COPY

UCID-20369

FEFFLAP: A Finite Element Program for Analysis of Fluid-Driven Fracture Propagation in Jointed Rock

Vol. 2: User's Manual and Model Verification

R. J. Shaffer
A. R. Ingraffea
F. E. Heuze

Prepared for
Morgantown Energy Technology Center
Morgantown, WV

March, 1985

Lawrence Livermore National Laboratory

Unconventional Gas Program

Western Gas Sands Research

INDEX

ABSTRACT

The stimulation of complex gas reservoirs is best done by massive fracturing. The fractures are driven by fluids such as gels and foams. The prediction of fracture extent requires very sophisticated models, to account for the real geologies in which induced fractures interact with natural discontinuities.

We have developed a state-of-the-art model to describe fluid-driven fracture propagation in naturally jointed, gas-bearing rock formations. It is a finite element code, named FEFFLAP (Finite Element Fracture and Flow Analysis Program). The program is highly interactive, with extensive graphical displays of the fracture behavior. Many automatic features for input generation, zoning, and rezoning make the code particularly efficient. The fracture mechanics, solid mechanics and fluid mechanics are fully coupled.

Model verification has been performed against analytical solutions and physical experiments. The program was developed on a CRAY computer and can be transcoded for use on workstations and minicomputers. This document constitutes the user's manual for the code and provides sample problems used for verification and demonstration of the code's versatility.

1. GENERAL DESCRIPTION OF FEFFLAP

1.1 Introduction

FEFFLAP (Finite Element Fracture and Flow Analysis Program) is a finite element program for two-dimensional analysis of static or quasi-static propagation of discrete fractures in homogeneous or jointed media. The fractures, or cracks, can be driven by a variety of loading conditions, including internal fluid pressure. The code contains quite sophisticated fracture mechanics: stress intensity factors are calculated with special crack tip finite elements; fracture instability and angle of propagation are estimated from any one of three fracture criteria; induced cracks can change direction and can interact with pre-existing fractures. FEFFLAP also provides for non-linear behavior of discontinuities such as geologic interfaces and joints, and for steady-state viscous fluid flow in the cracks and the discontinuities. Pressures and flow rates are determined from crack and joint apertures. These pressures are then used as boundary conditions for the structural analysis.

FEFFLAP has evolved from three building blocks:

- the FEFAP (Finite Element Fracture Analysis Program) for discrete fracture propagation in rock and concrete [1-3].
- the JTFLO code for coupled analysis of flow in fractured media [4], as further enhanced at LLNL.
- the JPLAXD code for static analysis of structures in jointed rock [5].

This document constitutes the user's manual for the CRAY version of FEFFLAP, numbered 1.0. A companion report* provides the theory and programming information. Because of the sophistication of the analyses which the code can perform, it is highly recommended that any potential user be familiar with the theory of FEFFLAP, rather than simply using the program as a black box.

---

*Ingraffea, A. R., Shaffer, R. J., and Heuze, F. E., "FEFFLAP: A Finite Element Program for Analysis of Fluid Driven Fracture Propagation in Jointed Rock - Vol. 1: Theory and Programmer's Manual", Lawrence Livermore National Laboratory, UCID-20368, 1985.

1.2 FEFFLAP Characteristics

1.2.1 Structural analysis aspects

The finite element library consists of seven elements of three types: solid, joint and flow elements. The flow element is the simplest; it is a line element with two nodes. The remaining six elements are shown in Figure 1. The joint (interface) element has six nodes and a prescribed thickness. The five solid elements range from a truss element with three nodes to the quadrilateral with eight nodes. Singular elements as shown in Figure 1 e) and f), are used only at crack tips and are designed to respond to the inverse square root singularity in the stresses, for the calculation of stress intensity factors [6,7]. The code can accommodate up to 10 different materials and can propagate up to 9 cracks at a time; these numbers can be increased with minimal changes.

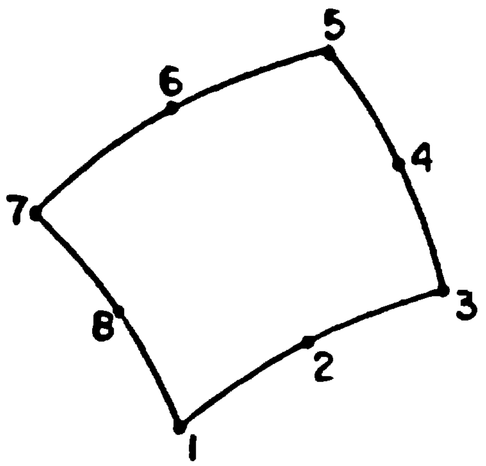
The finite element formulation is based on the displacement approach. The structural stiffness matrix is obtained by direct assembly of elemental stiffnesses. Automatic mesh generation and bandwidth minimization routines are available. The system of equation is solved directly by a highly efficient skyline solver [8]. The structural part of FEFFLAP accepts three types of loading conditions:

- point loads
- edge loads
- initial nodal displacements.

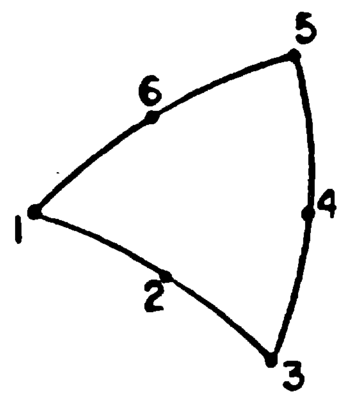
In addition, the fluid flow part of FEFFLAP uses two types of boundary conditions: pressure and flow rate. These are prescribed at the flow nodes.

The program is both incremental and iterative. Incremental means the ability to apply loads in a stepwise fashion. This emulates to some degree the time dependency of a loading, such as a borehole being pressurized in rock; however, the solution at each load increment is steady state, not transient. Iterative means that the calculations are performed repeatedly within each load increment to satisfy the non-linear constitutive relations of geological materials, and the coupling between the flow and the structural stress changes.

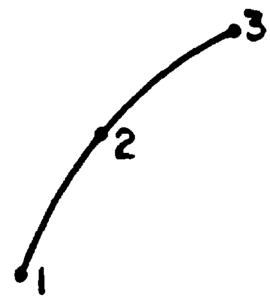
FEFFLAP can calculate the exact load required to initiate or extend any crack. A multiple load capability is available in the code, for complicated



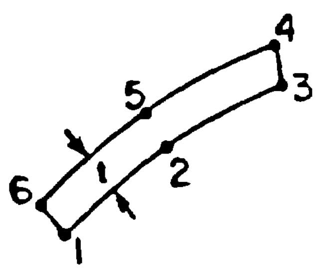
a) Quadrilateral (Q8)



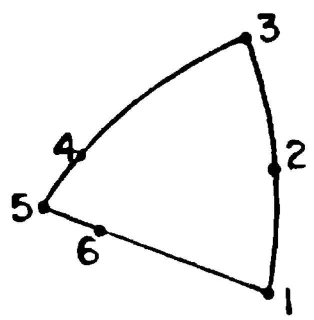
b) Triangle (LST)



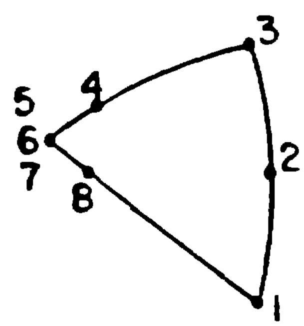
c) Truss



d) Interface



e) Singular LST



f) Singular (collapsed) Q8

Figure 1: FEFFLAP Element Library

fracturing problems involving simultaneously loads that can change and loads that do not change. An example of such a problem is hydraulic fracturing of a gas field from a borehole: the field stresses at infinity are constant in time during fracture propagation, and the fluid pressure in the borehole is variable; inside the domain, however, the field (in-situ) stresses can vary arbitrarily in space. The multiple load vector capability is described in detail in the theory manual.

1.2.2 Fracture mechanics logic

The sequence of events in the implementation of linear elastic fracture mechanics (LEFM) into FEFFLAP is:

(1) Compute stress intensity factors for present crack tip location and loading.

(2) Substitute K_I and K_{II} into any of three [9-11] mixed-mode interaction formulas. Compute new crack direction and assess stability. If crack is unstable, continue. If stable, go to step 4.

(3) Remesh for a selected increment of propagation. Repeat steps 1 through 3 until crack is stable or fracture occurs.

(4) If crack is stable, raise load level until instability is predicted by interaction formula. Continue with step 3.

A nonlinear fracture mechanics capability also exists in FEFFLAP. Further details for both types of analyses can be found in the theory manual.

1.2.3 Mesh modification - propagating cracks

The strategy for crack extension through a finite element mesh is as follows:

(1) From the initial direction of the crack axis and the predicted angle of crack extension, determine the incipient angle of crack propagation in global coordinates.

(2) Move the quarter-point nodes back to their initial midside positions, to remove the local singularity from the old crack tip location.

(3) Define a new crack tip node whose coordinates are determined from the crack length and angle of crack extension.

(4) Define a new node adjacent to the old crack tip node, to provide for crack opening/closing behind the tip.

(5) Search the previous singular elements, to determine which one is going to be crossed by the crack.

(6) If the new crack tip node falls inside this element, extend the crack to it and go to 8; otherwise, simply extend the crack through the entire length of the element.

(7) Locate the next element to be crossed by the crack and go to 6.

(8) Define the new nodes from which the stress intensity factors will be evaluated.

(9) Adjust the midside nodes to the quarter-point position, for elements around the new crack tip.

(10) Display the modified mesh, to allow the user to interactively perform final adjustments.

1.2.4 <u>Coupling of flow and structural analyses</u>

The solid part of FEFFLAP is coupled with a flow program. Structural displacements under all the loads, including fluid pressures, determine the apertures of the joints and cracks. The apertures are then used in the fluid flow model to determine flow rates and pressures. This process is repeated until convergence occurs. The nonlinear behavior of joints is also iterated upon until convergence is achieved within the flow iteration loop. The logic of the flow/structure coupling is described in Figure 2.

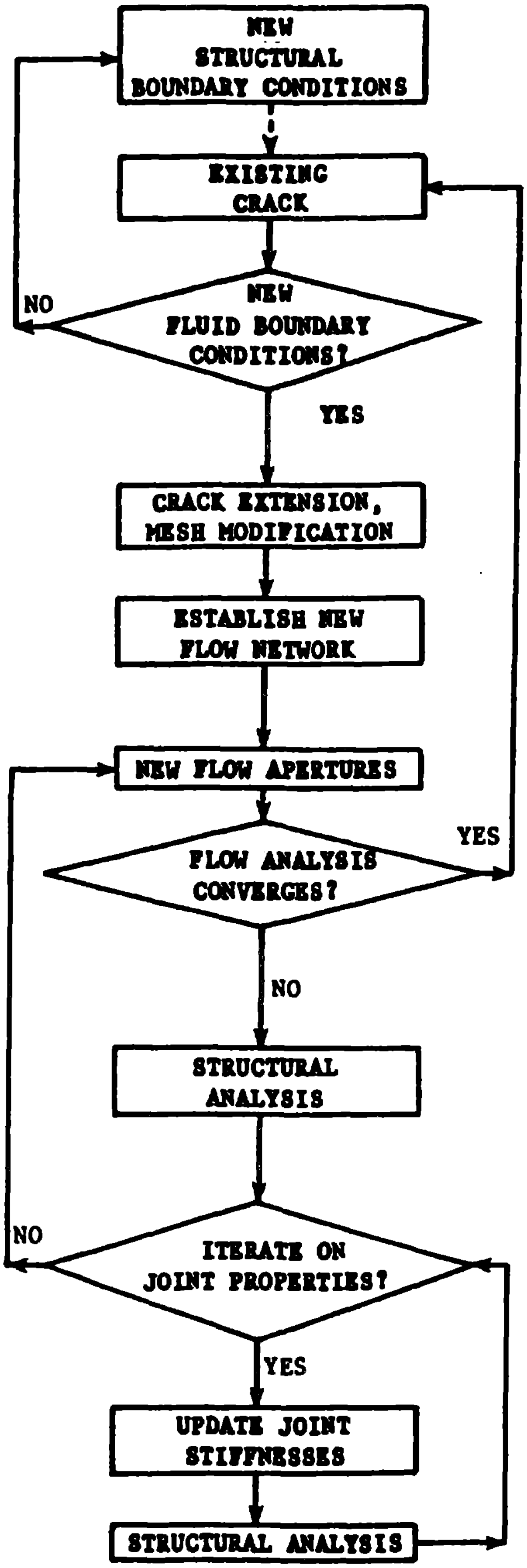


Figure 2. Flow Chart of FEFFLAP

1.2.5 Graphics and interactivity

The operation of the code depends heavily on graphics because it is a fully interactive model. The user can alter crack propagation by changing conditions during the analysis, can iterate on fluid flow and joint nonlinearities, as required for convergence, and can change aspect ratios of elements at each increment of crack propagation, to improve accuracy. Each analysis step is directed by the user with alphanumerical and graphical feedback of the results of each step. The program suggests courses of action, but these may be overridden by the user. After any complete crack propagation step, the analysis can be terminated and easily restarted from the previous step. The emphasis in the program design is on providing versatility to the researcher: he is not locked into a batch-produced result via the initial data input.

Operation of FEFFLAP requires an external graphics routine library, the PLOT10 Terminal Control System, Release 3.0, which is produced by Tektronix, Inc. These routines in turn require a graphics terminal of the Tektronix 4000 series or an emulator of such.

1.2.6 Post processing

Three types of output are provided to the user: monitor graphics, teletype output, and accumulated output that is put into a file named by the user. On the CRAY computer the monitor graphics can be accumulated into a graphics file if desired, and hardcopy can then be obtained after execution of the program. In addition, FILE1 contains the values of all the variables after the last flow and joint iterations, or the last crack increment. Therefore, additional processing could be done if desired.

1.2.7 Program size

FEFFLAP is written in Fortran IV, with about 21,000 lines organized into about 100 subroutines. There are no direct limits on the initial number of nodes or elements as the program was developed for virtual memory operating systems. However, the size of the blank common (MTOT in subroutine MAIN) should be adjusted according to the machine size made available to the user.

2. PREPARATION OF INPUT DATA

2.1 Input Format and Instructions

```
+-----------------------------------------------------+
|   CARD SET 1                                        |
|   TITLE CARD                                        |
|   FORMAT : I2,9A8                                   |
|   NUMBER OF CARDS IN SET : 1                        |
+-----------------------------------------------------+
```

NOTES	COLUMNS	VARIABLES	ENTRY
1	3-75	TITLE	Title of the problem, limited to 72 alphanumeric characters

NOTES:

1) Note that the title card begins in the 3rd column. See note 8 of card set 2.

```
+-----------------------------------------------------+
|   CARD SET 2                                        |
|   CONTROL CARD                                      |
|   FORMAT : 8I5,14I1                                 |
|   NUMBER OF CARDS IN SET : 1                        |
+-----------------------------------------------------+
```

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	NPOIN	Total number of nodes
12	6-10	NELEM	Total number of elements
	11-15	NVFIX	Total number of restrained boundary nodes, where one or more degrees of freedom are restrained
1,2,3	16-20	NMATS	Total number of material types
	21-25	NGAUS	Order of stiffness integration formula for numerical integration (generally use 2)
10	26-30	NTYPE	Problem type parameters: 0 Axisymmetric 1 Plane stress 2 Plane strain
4,5	31-35	NCRAC	Initial number of crack tips
13	36-40	NUMLDSET	Number of load sets
6	41	IMODE(1)	Print input data (0=no, 1=yes, 2=perform mesh optimization and print data)

Notes	Column	Variable	Description
	42	IMODE(2)	Plot initial mesh (0=no, 1=yes
	43	IMODE(3)	Program execution mode (0=data check, 1=problem solution)
	44	IMODE(4)	Compute principal stresses (0=no, 1=yes)
	45	IMODE(5)	Plot deformed mesh (0=no, 1=yes)
7	46	IMODE(6)	Plot principal stress vectors (0=no, 1=yes)
8	47	IMODE(7)	Save results on unit 1 (0=no, 1=yes)
(presently inactive)	48	IMODE(8)	Update stiffness matrix of concrete elements based on a triaxial stress-strain model OTTOSEN [12,13], (0=no, 1=yes)
2,3	49	IMODE(9)	Number of interface material types
	50	IMODE(10)	Read and plot experimental load deflection curve (0=no, 1=yes)
11	51	IMODE(11)	Strains in interface element are to be divided by its in plane thickness (t in Fig. 1.d). (0=no, 1=yes)
	52	IMODE(12)	Compute reactions (0=no,1=yes)
9	53	IMODE(13)	Element stiffness matrices already computed and stored in unit 8 (0=no, 1=yes)
14,15	54	IMODE(14)	Pressurize cracks and/or interfaces. 0 = no pressurization 1,2,3 = fluid flow analysis in cracks and interfaces 4 = constant pressure in cracks only

NOTES:

1) Interface elements must have their own material number.
2) At most 5 interface element material types can be specified.
3) Interface element material types are numbered last.
4) If crack propagation analysis is to be performed, NCRAC should be the negative of the initial number of crack tips.
5) If crack propagation analysis is to be performed with no initial crack tips, NCRAC should be 9.
6) If mesh optimization is in effect, and problem is not be to executed, optimized mesh data are written on unit 7 and a table of the new/old nodes, and element numbers appears in the output. Nodal coordinates, nodal connectivity, boundary conditions, load vectors, and all appropriate data are automatically updated to reflect the renumbering scheme.
7) Principal stresses are not plotted in singular elements or in interface elements.
8) If crack propagation analysis is to be performed and IMODE(7) is not equal to zero, all the data will be dumped on unit 1 after each crack increment; to perform a reanalysis, the number 99 should be inserted into the first 2 columns of the title card of card set 1.

9) If a reanalysis (without crack propagation) is to be reperformed without alterations affecting the global stiffness matrix, substantial CPU time can be saved by avoiding the reevaluation of the elements stiffness matrices.

10) For axisymmetric analysis, the y axis corresponds to the axis of symmetry (z) and the x axis is along the radial direction (r).

11) For default interface element models, Card Set 8.2, IMODE(11) must be zero.

12) The program is currently dimensioned for up to 100 interface elements. This dimension on variables ENJNT (100,3), ESJNT (100,3), and NUMJNT (100,3) must be changed to accommodate more interface elements.

13) Analysis involving non-proportional loading can be made by using two load sets. The first load set consists of all loads to be held constant during an analysis. All variable loads are input in the second load set. All loads in the second set are proportionally modified interactively during analysis

14) A coupled, steady-state fracture/flow analysis will be performed if IMODE(14) = 1, 2, or 3. In this case KAGR, Card Set 6.3.1, must be non-zero. See Card Set 10.

15) If IMODE(14) = 4, a uniform pressure will be applied in cracks only. KAGR must be zero, Card Set 6.3.1. See Card Set 9.3.6.

CARD SET 2.1
CONTROL CARD
FORMAT : I1
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
1	1	IMODE(15)	0 = do not iterate on interface element stiffnesses 1 = iterate on interface stiffnesses

NOTES:

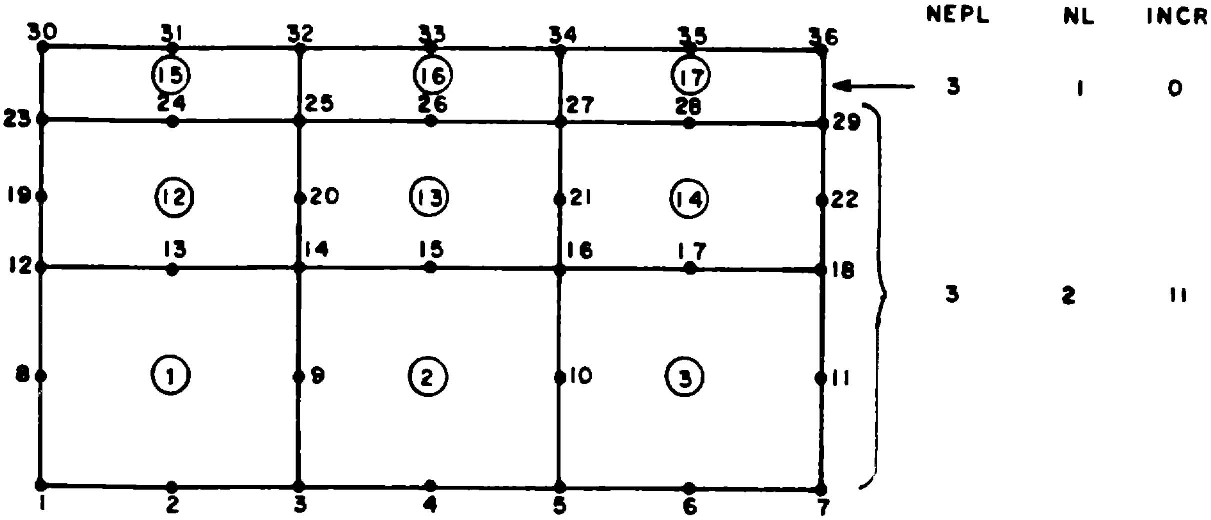
1) The purpose of this card is to give the user the option of performing analysis involving nonlinear interface properties, and to interactively modify the load factor. See Card Set 8.2.

CARD SET 3
ELEMENT CARDS
FORMAT : 13I5
NUMBER OF CARDS IN SET : Variable

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	NUM	Element number
	6-10	MATNO(NUM)	Material type number
1,2,3,9	11-15	LNODS(NUM,1)	First node number
4,5	16-20	LNODS(NUM,2)	Second node number
	---	-- -- --	-- -- -- -- -- -- -- --
	46-50	LNODS(NUM,8)	Eighth node number
6,7	51-55	NEPL	Number of elements to be generated per layer
	56-60	NL	Number of layers of elements to be generated
8	61-65	INCR	Increment in element number between two adjacent elements in subsequent layers

NOTES:

1) Node number must be listed in a counterclockwise sequence, starting from any corner node.
2) For triangular or interface elements, only the first 6 nodes need be specified.
3) For a truss element, only the first 3 nodes need be specified.
4) Collapsed quadrilateral element should have nodes 4, 5, and 6 corresponding to the crack tip node.
5) For singular triangular element, node numbering should start at side opposite to crack tip so that crack tip node is fifth node number.
6) For element generation, numbering of nodes must be along the layers (Fig. 3).
7) Generation begins with the current element card; 2 consecutive elements will have their sides 2 and 4 in common.
8) Default value of INCR is NEPL.
9) Input first node number at lower righthand node of element for generation of Q8 elements when element layers are parallel to the y-axis.



NUM	MATNO (NUM)	LNODS								NEPL	NL	INCR
1	1	1	2	3	9	14	13	12	8	3	2	11
15	2	23	24	25	32	31	30	0	0	3	1	0

Cord set 3

Figure 3. Element Generation Examples

CARD SET 4
NODE CARDS
FORMAT : I5, 2F10.0, 3I5, 2F10.0
NUMBER OF CARDS IN SET : Variable

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	INOD	Node number
1,2	6-15	COORD(INOD,1)	X-Coordinate of INOD
	16-25	COORD(INOD,2)	Y-Coordinate of INOD
3,4	26-30	IA	Axis of node generation (Fig. 4): 1 Block generation in x direction 2 Block generation in y direction 3 Row generation in arbitrary direction 4 Circular arc generation (with the x-axis not being crossed during the generation, or if last node generated is on the x-axis) 5 Circular arc generation (with the x-axis being crossed during the generation)
	31-35	NEPL	Number of elements per layer whose nodes are to be generated
	36-40	NL	Number of layers of elements whose nodes are to be generated
	41-50	XCEN	X coordinate of fictitious center point (if IA=4 or 5)
	51-60	YCEN	Y coordinate of fictitious center point (if IA=4 or 5)

NOTES:

1) The coordinates of the last node have to be input and not generated.
2) Midside nodes are automatically generated (by linear interpolation) unless they are attached to interface elements only.
3) Generation is performed with respect to the previously defined node.
4) For IA=4 or 5, generation is CW only.

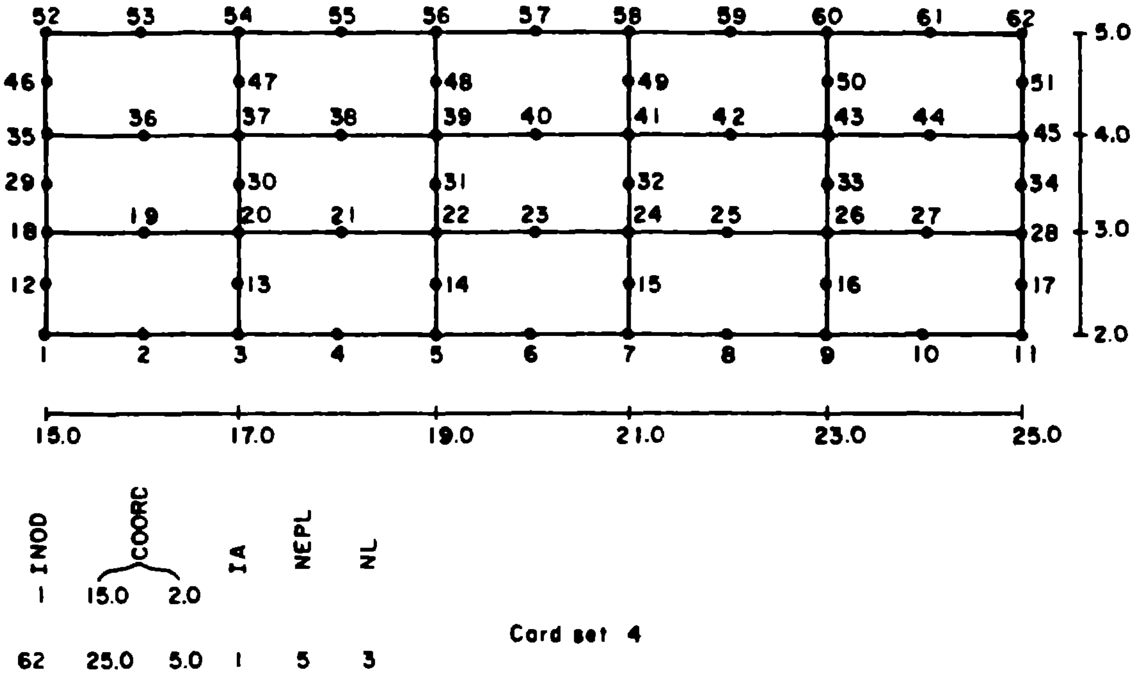


Figure 4a. Example of Block Nodal Generation

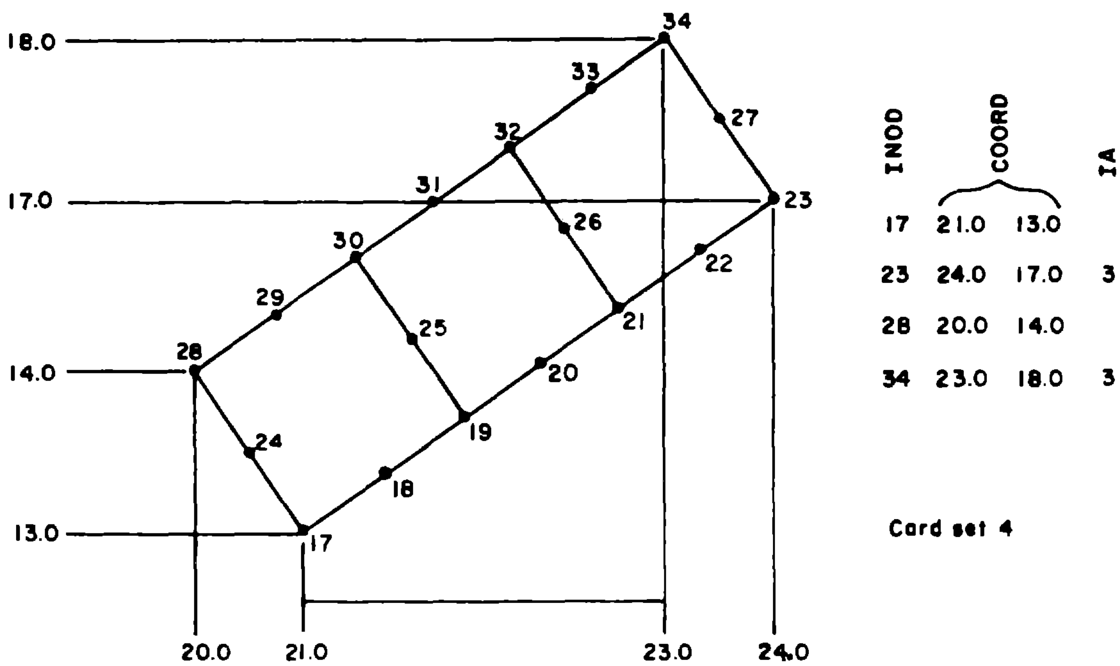
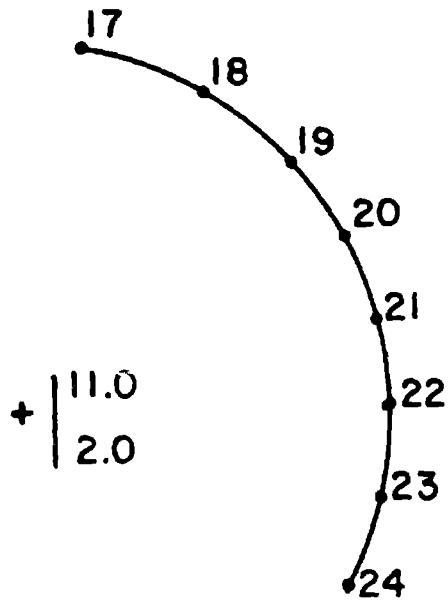


Figure 4b. Example of Row Nodal Generation



INOD	COORD		IA	NEPL	NL	XCEN	YCEN
17	11.0	5.0					
24	15.0	−1.0	5	0	0	11.0	2.0

Card set 4

Figure 4c. Example of Circular Nodal Generation

CARD SET 5
Only if IMODE(2), IMODE(5), or IMODE(6)=1 in Card 2
PLOTTING CONTROL CARD
NUMBER OF CARDS IN SET : 2 or 4

CARD SET 5.1
FORMAT : 2I5
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
1	1-5	IBAUD	Transmission (BAUD) rate in characters per second
	6-10	JTEKNB	Tektronix terminal type used (4000 series)

NOTES:

1) Check BAUD rate at rear of TEKTRONIX terminal used.

CARD SET 5.2
FORMAT : 2I5, 4F10.0
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	LABELE	Label element numbers (0=no, 1=yes)
	6-10	LABELN	Label node numbers (0=no, 1=yes)
	11-20	XMIN	Minimum value of X in the plot
	21-30	XMAX	Maximum value of X in the plot
	31-50	YMIN	Minimun value of Y in the plot
	41-50	YMAX	Maximum value of Y in the plot

CARD SET 5.3
FORMAT : I5
Only if IMODE(6)=1 in Card 2
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	JSTRSG	Sign of principal stresses to be plotted; 1=positive only, 2=positive and negative (tension is positive)

CARD SET 5.4
FORMAT : 16I5
NUMBER OF CARDS IN SET : 1
Only if IMODE(6)=1 in Card 2

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	IBPS(1)	First node defining region inside which stresses are to be plotted
	5-10	IBPS(2)	Second node defining region inside which stresses are to be plotted
	10-15	etc...	

CARD SET 6
CRACK ANALYSIS INFORMATION
Only if NCRAC NE 0 in Card 2
NUMBER OF CARDS IN SET : Variable

CARD SET 6.1
CRACK TIP INFORMATION
Only if NCRAC≠9 or 0.
FORMAT : 14I5
NUMBER OF CARDS IN SET : |NCRAC|

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	ICRAC	Crack number
1,2	6-10	LCRAC(ICRAC,1)	Crack tip node number (Node A)
	11-15	LCRAC(ICRAC,2)	Node B
	16-20	LCRAC(ICRAC,3)	Node C
	21-25	LCRAC(ICRAC,4)	Node D
	26-30	LCRAC(ICRAC,5)	Node E
	31-35	NECRAC(ICRAC,1)	First element around crack tip
	36-40	NECRAC(ICRAC,2)	Second element around crack tip
	-----	-----	------------------------------
	65-70	NECRAC(ICRAC,8)	Eighth element around crack tip

NOTES:
1) See Fig. 5. For a crack tip along a symmetry line, only 3 nodes have to be defined. The positive crack axis (x) must coincide with the positive global X or Y axis.
2) Note that the end of a discontinuous joint must be treated as a crack tip, if extension of this end is possible.

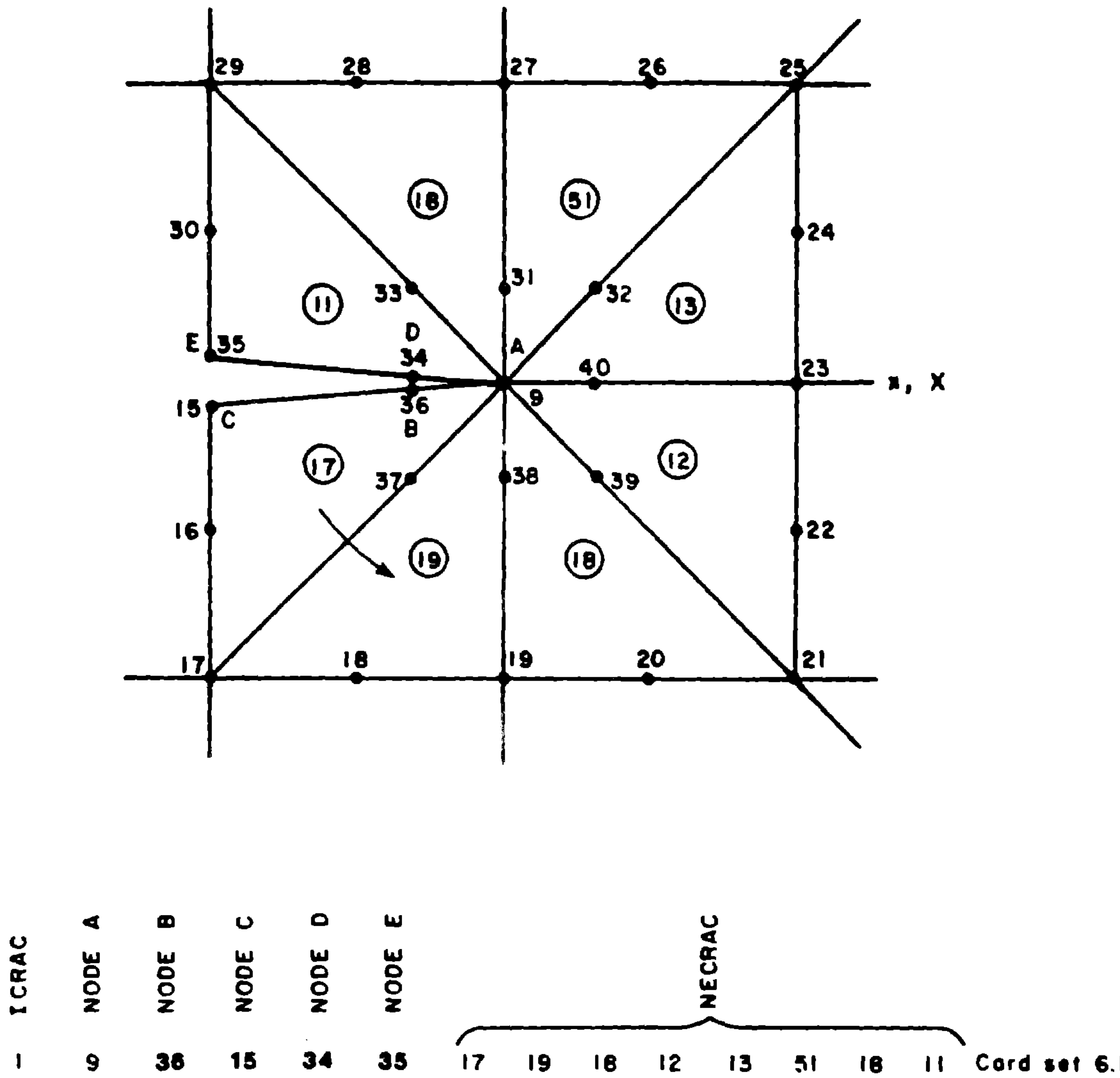


Figure 5. Example of Crack-Tip Information

CARD SET 6.2
FRACTURE ANALYSIS CONTROL CARD
FORMAT : 2I5
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	IFRAN	Mode of fracture analysis 1=Computation of stress intensity factors only 2=Computation of stress intensity factors, and determination of angle of crack propagation 3=Full quasi-static or unstable crack propagation analysis with self-modification of the mesh
6-10		ITEORY	Mixed-mode fracture criterion to be used: 1 Sigma theta max 2 S theta min 3 G theta max

---

CARD SET 6.3
Only if IFRAN=3 in Card 6.2
CRACK EXTENSION INFORMATION
NUMBER OF CARDS IN SET : 3

CARD SET 6.3.1
FORMAT : 3I5
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	JNWCRC	Number of allowable crack tips
	6-10	MAXCRA	Number of allowable crack increments
	11-15	KAGR	Material property number of interface elements to be inserted along propagating crack, (KAGR=0 for no interface elements)

CARD SET 6.3.2
FORMAT : 8F10.0
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
	1-10	CRAINC	Length of each crack increment
1	11-20	THCKRK	Thickness of propagating cracks
	21-30	ANGMAX	Maximum angle (degrees) to be sustained by crack tip elements (should be 40-60 degrees)
2	31-40	RSLOPE	Slope of the R curve
	41-50	FC	Uniaxial compressive strength of material 1
	51-60	FT	Uniaxial tensile strength of material 1
	61-70	TENCOM	Tensile stress causing crack opening in tensile-compressive region of material 1

NOTES:

1) This feature is essentially for graphic display, but it will also modify slightly the stiffness matrices of the adjoining elements.
2) The zero intercept of the R curve will be computed by the program in terms of CK1 and of the elastic properties of material number 1.

CARD SET 6.3.3
FORMAT : 3I5
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	NODMON(1)	First node of which displacements are to be monitored
	6-10	NODMON(2)	Second node of which displacements are to be monitored
	11-15	IWMON	Degree-of-freedom (DOF) to be monitored 0 = X-displacement 1 = Y-displacement

NOTES:

1) The DOF defined by IWMON of NODMON(1) will be used in a plot of load factor versus displacement.

CARD SET 6.4
PLOTTING CONTROL CARDS
Only if Card Set 5.1 has not been defined
FORMAT : 2I5
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	IBAUD	Transmission (BAUD) rate in characters per second,
	6-10	JTEKNB	Tektronix terminal number used (4000 series)

CARD SET 6.5
CRACK PLOTTING CARD
FORMAT : 16I5
Only if IFRAN=3 or 4
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	ICBP(1)	First node defining region inside which cracks are to be plotted
	6-10	ICBP(2)	Second node defining region inside which cracks are to be plotted
	----	-----	------------------------------

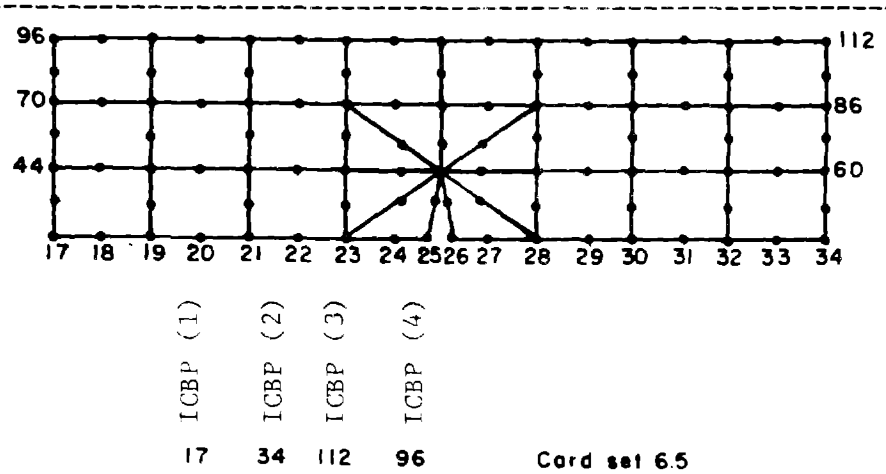


Figure 6: Example of Region Definition.

CARD SET 6.6
GENERAL CRACK GEOMETRY
FORMAT : 16I5
NUMBER OF CARDS IN SET : 1 for each crack tip
Only if IFRAN=3 or 4, and NCRAC NE to 0 or 9

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	ICRAC	Crack tip number
	6-10	KRKTAL	Crack tip number attached to the tail of crack ICRAC; 0 if none (crack originates from a free surface)
1	11-15	KRAHIS(ICRAC,1)	First node along crack ICRAC
	16-20	KRAHIS(ICRAC,2)	Second node along crack ICRAC
	-----	--------------	----------------------------

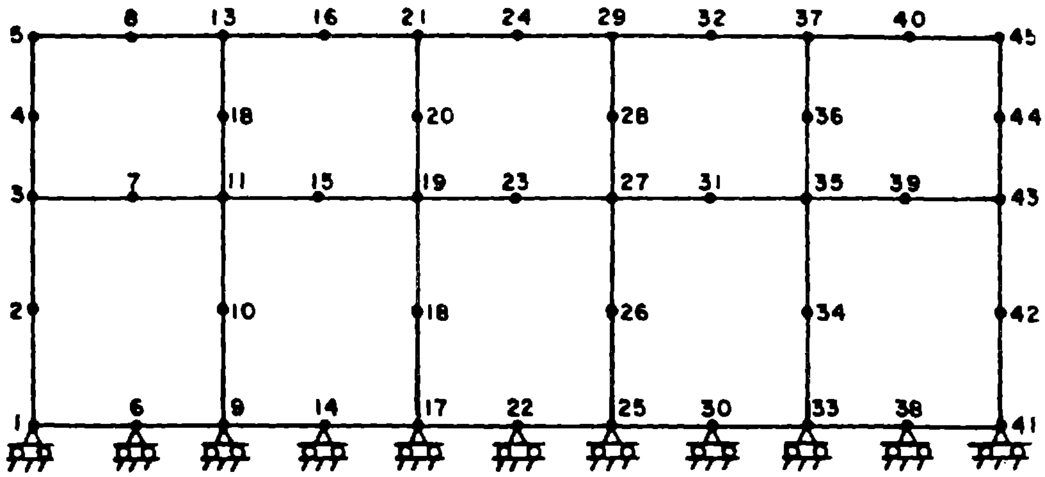
NOTES:

1) Start at tail of crack; input every node along one side of the crack (used for plotting purposes only).

---------------+-------- ---------

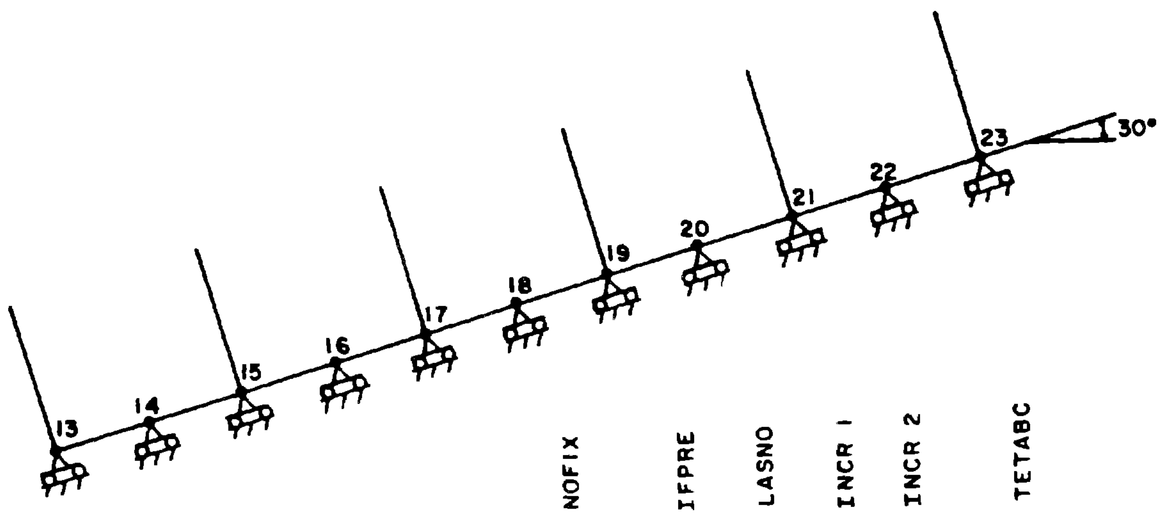
CARD SET 7
RESTRAINED NODES CARDS (see examples on Fig. 7)
FORMAT : I5, 3X, 2I1, 3I5, F10.0
NUMBER OF CARDS IN SET : Variable

NOTES	COLUMNS	VARIABLES	ENTRY
1,2	1-5	NOFIX(IVFIX)	Restrained node number
	9	IFPRE(IXFIX,1)	Condition of restraint on X displacement 0=no restraint 1=restraint
	10	IFPRE(IVFIX,2)	Condition of restraint on Y displacement 0=no restraint 1=restraint
3	11-15	LASNO	Last node number on a row B.C. (Boundary Condition) generation
	16-20	INCR1	Difference between midside node number and corner node number along line of B.C. generation
	21-25	INCR2	Difference between corner node number and midside node number along line of B.C. generation
4	26-35	TETABC	Transformation angle for inclined B.C. restrained in the Y direction



NOFIX	IFPRE	LASNO	INCR 1	INCR 2	
1	01	41	5	3	Card set 7

Figure 7a. Example of Boundary Condition Generation



NOFIX	IFPRE	LASNO	INCR 1	INCR 2	TETABC
13	00	0	0	0	30.0
⋮					
23	00	0	0	0	30.0

(b) Card set 7

Figure 7b. Example of Inclined Boundary Condition Specification (no generation possible in this case)

NOTES:

1) In axisymmetric analyses, all the nodes along the axis of revolution (Y) should be restrained against displacements in the X direction.
2) Nodes with an initial displacement should be restrained in that particular direction.
3) Generation for B.C. nodes should start and end at an element corner node.
4) For inclined B.C. data, no generation is possible; at most 50 nodes with identical TETABC can be specified, and IFPRE(1) and IFPRE(2) are to be left equal to 0.

CARD SET 8
MATERIAL TYPE CARDS
NUMBER OF CARDS IN SET : NMATS, from Card 2

CARD SET 8.1
MATERIAL CHARACTERISTICS
FORMAT : I5, 6F10.0
NUMBER OF CARDS IN SET : NMATS-IMODE(9), from card 2

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	MAT	Material type number
	6-15	PROPS(MAT,1)	Young's modulus
1	16-25	PROPS(MAT,2)	Poisson's ratio
2	26-35	PROPS(MAT,3)	Material thickness in direction normal to the plane of analysis
	36-45	PROPS(MAT,4)	Mass density
	46-55	PROPS(MAT,5)	Coefficient of thermal expansion
	56-65	PROPS(MAT,6)	K_{Ic}, plane strain fracture toughness

NOTES:

1) If material type number is one of a truss element, PROPS(MAT,2) is the yield stress and PROPS(MAT,3) is the cross sectional area.
2) PROPS(MAT,3) is defaulted to unity.

CARD SET 8.2
INTERFACE ELEMENT CONTROL DATA
Only if IMODE(9) NE 0, from Card 2

CARD SET 8.2.1
FORMAT : I2, I3, 7F10.0
NUMBER OF CARDS IN SET : IMODE(9)

NOTES	COLUMNS	VARIABLES	ENTRY
1,4,5	1-2	NJOINM	Interface material number
	3-5	NTRFTP	Interface type:
7			Closure Model 1: Normal stiffness will be assigned if the faces of the interface element have a negative relative displacement (takes into account only the displacements)
8			Closure Model 2: Normal stiffness will be assigned if the 2 faces of the interface element will overlap (takes into consideration both coordinates and displacements)
2			Shear Model 3: Aggregate interlock model for concrete
9	6-15	ENORMJ	Initial interface normal stiffness
	16-25	ESHEAR	Initial interface shear stiffness
6	26-35	COHES	Joint peak cohesion
3	36-45	CINIT	Initial crack opening for crack experiencing aggregate interlock
5	46-55	PHIR	Residual friction angle, in degree
6,5	56-65	STRPOS	Tensile strength
5	66-75	CLOMAX	Maximum joint closure under compression, input as a positive number. Defaulted to 0.00 * element length.

NOTES:

1) The interface element material models must be listed in order (i.e., the first card of this set must have NJOINM equal to the lowest interface material number.
2) If NTRFTP = 3, the program will assign to interface elements inserted in a crack in concrete the shear stiffness derived by Fenwick and Paulay [14] which is inversely proportional to the crack opening. All problem units must be in kips and inches to use this particular model. If crack is tending to open, or if open and tending to close, normal stiffness will be set to zero. If crack sides come into contact on closure, normal stiffness will be automatically set to high value to prevent overlap. The only additional variable required on this card if NTRFTP=3 is CINIT. If NTRFTP=3, the concrete elements must have material property set 1 (MAT=1 in Card Set 8.1, card 1).
3) CINIT, the initial crack width, is used only when NTRFTP=3. CINIT must be > 0.001 in.
4) The out-of-plane interface thickness is the same as the thickness of material number 1.
5) A nonlinear joint model described in Figures 8 to 11 is available. See Note 6. User must re-program subroutine DJOINT for other nonlinear interface models. Refer to comments in DJOINT for further details.
6) If user specifies finite values for STRPOS or COHES, i.e., if shear or tensile failure modeling is desired, IMODE(16) on Card Set 2.1 should be equal to 1 so that iterations can be performed until user is satisfied that convergence has occurred.
7) This model is typically used for a rock joint with filling material. Non-zero values of ENORMJ, ESHEAR, and CLOMAX should be input (Figures 9, 10).
8) This model is typically used for crack interface elements, gaps, or joints without filling material. All properties should be set to zero except PHIR which becomes the friction angle between two potentially contacting materials.
9) Assumes the same value for opening and closing. See Notes 7 and 8.

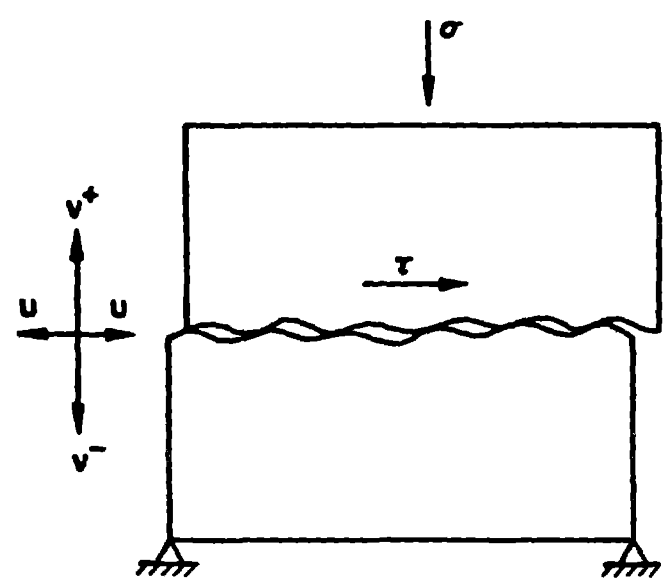


Figure 8. Nomenclature for Joints in Direct Shear

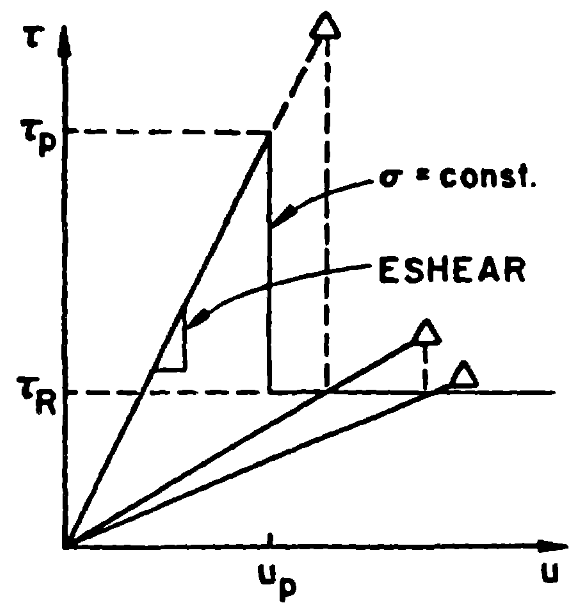


Figure 9. Strain-Softening of Joints in Direct Shear

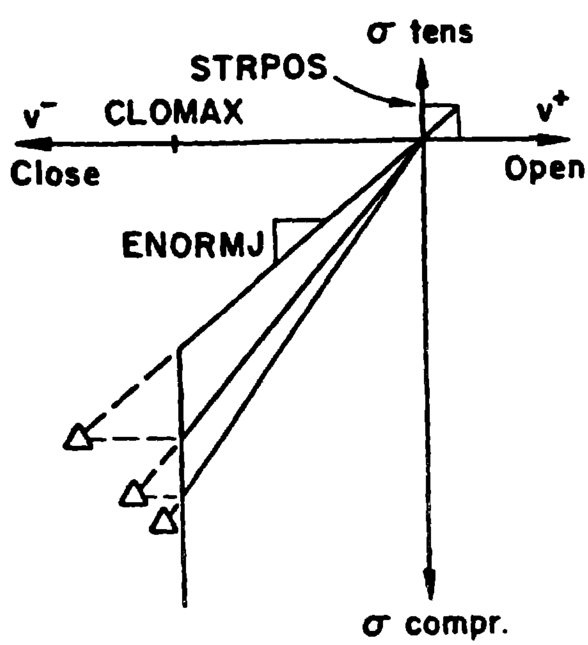


Figure 10. Joint Behavior in the Normal Direction

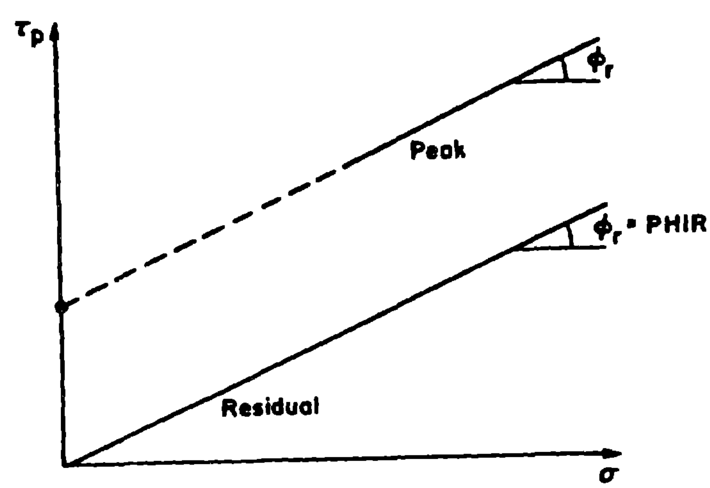


Figure 11. Peak and Residual Joint Shear Strength Envelopes

CARD SET 8.3
Experimental load deflection curve
Only if IMODE(10)=1
NUMBER OF CARDS IN SET : Variable

CARD SET 8.3.1
FORMAT : I5
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	NPDEL	Number of points defining the experimental load deflection curve (fewer than 51)

CARD SET 8.3.2
FORMAT : 2F10.0
NUMBER OF CARDS IN SET : NPDEL

NOTES	COLUMNS	VARIABLES	ENTRY
	1-10	LOAD	Value of the load
	11-20	DELTA	Deflection corresponding to load

CARD SET 9
LOAD DATA
This set is repeated once for each load set.
NUMBER OF CARDS IN SET : Variable

CARD SET 9.1
LOAD TITLE
FORMAT : 12 A6

NOTES	COLUMNS	VARIABLES	ENTRY
	1-72	TITLE	Title of the load case; limited to 72 alphanumeric characters

CARD SET 9.2
LOAD DATA CONTROL CARD
FORMAT : 5I5

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	IPLOD	Applied point load control parameter: 0 = No applied nodal loads to be input; 1 = Applied nodal loads to be input
	6-10	IGRAV	Gravity loading control parameter: 0 = No gravity loads to be considered; 1 = Gravity loads to be considered
	11-15	IEDGE	Distribution edge load control parameter: 0 = No distributed edge loads to be input; 1 = Distributed edge loads to be input
	16-20	ITEMP	Thermal loading control parameter: 0 = No thermal loading to be considered; 1 = Thermal loading to be considered
	21-25	INIDIS	Initial displacement control parameter: 0 = No initial displacements; 1 = Initial displacements

NOTE: In the current version of FEFFLAP (1.0) the gravity and thermal loadings are operational only when no crack propagation takes place. i.e. when no new nodes are generated during the analysis.

CARD SET 9.3.1
APPLIED LOAD CARDS
Only if IPLOD = 1
NUMBER OF CARDS IN SET : One card for each loaded node (+1 if last node not loaded)

NOTES	COLUMNS	VARIABLES	ENTRY
1	1-5	LODPT	Node number
	6-15	POINT(1)	Load component in X direction
	16-25	POINT(2)	Load component in Y direction

NOTES:

1) This last card should be that for the highest numbered node whether it is point loaded or not.

CARD SET 9.3.2
GRAVITY LOADING CARD
Only if IGRAV=1
FORMAT : 2F10.0
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
	1-10	THETA	Angle of gravity axis from the positive Y axis
	11-20	GRAVY	Gravity constant-specified as a multiple of the gravitational acceleration, g

---

CARD SET 9.3.3 DISTRIBUTED EDGE LOAD CONTROL CARD Only if IEDGE=1 NUMBER OF CARDS IN SET : Variable

CARD SET 9.3.3.1 FORMAT : I5 NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
1	1-5	NEDGE	Number of element edges on which distributed loads are to be applied

NOTES:

1) Subsets 9.3.3.2 and 9.3.3.3 must be repeated in turn for every loaded element.

CARD SET 9.3.3.2 FORMAT : 4I5 NUMBER OF CARDS IN SET : NEDGE

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	NEASS	The element number with which the element edge is associated
	6-10	NOPRS(1)	List of nodal points, in a counterclockwise sequence,
	11-15	NOPRS(2)	of the nodes forming the element face on which the
	16-20	NOPRS(3)	distributed load acts

CARD SET 9.3.3.3
FORMAT : 6F10.0
NUMBER OF CARDS IN SET : NEDGE

NOTES	COLUMNS	VARIABLES	ENTRY
1	1-10	PRESS(1,1)	Value of normal component of distributed load at node NOPRS(1)
2,4,5	11-20	PRESS(2,1)	Value of normal component of distributed load at node NOPRS(2)
	21-30	PRESS(3,1)	Value of normal component of distributed load at node NOPRS(3)
3,4,5	31-40	PRESS(1,2)	Value of tangential component of distributed load at node NOPRS(1)
	41-50	PRESS(2,2)	Value of tangential component of distributed load at node NOPRS(2)
	51-60	PRESS(3,2)	Value of tangential component of distributed load at node NOPRS(3)

NOTES:

1) Subsets 9.3.3.2 and 9.3.3.3 must be repeated in turn for every element edge on which a distributed load acts. The element edges can be considered in any order.
2) A pressure normal to a face is assumed to be positive if it acts towards the inside of an element
3) A tangential load is assumed to be positive if it acts in a counter-clockwise direction with respect to the loaded element.
4) If the thickness of NEASS is different than 1., then the user should multiply the actual pressure load (force/area) by the thickness and input the distributed load as force/length.
5) In axisymmetric analysis, the distributed load is to be kept in terms of load/area and an analysis of a 1 radian segment will be performed.

```
+--------------------------------------------------------+
|   CARD SET 9.3.4                                       |
|   Only if ITEMP=1, in Card 9.2                         |
|   FORMAT : I5, F10.0                                   |
|   NUMBER OF CARDS IN SET : Variable                    |
+--------------------------------------------------------+
```

NOTES	COLUMNS	VARIABLES	ENTRY
1,2	1-5	NODPT	Node number
	6-15	TEMPE(NODPT)	Temperature at node

NOTES:

1) Datum temperature is taken to be zero.
2) Only nodal temperatures which are non-zero need be input. The card set must terminate with the highest numbered node regardless of the temperature value at this node.

NOTE: Thermal loading is operational only in the absence of crack propagation. See NOTE at 9.2.

---------------+---------------

```
+--------------------------------------------------------+
|   CARD SET 9.3.5                                       |
|   INITIAL DISPLACEMENT CARDS                           |
|   Only if INIDIS=1, in Card 9.2                        |
+--------------------------------------------------------+
```

---------------+---------------

```
+--------------------------------------------------------+
|   CARD SET 9.3.5.1                                     |
|   INITIAL DISPLACEMENT CONTROL CARDS                   |
|   FORMAT : I5                                          |
|   NUMBER OF CARDS IN SET:  Variable                    |
+--------------------------------------------------------+
```

NOTES	COLUMNS	VARIABLES	ENTRY
1,2	1-5	NINID	Number of elements having initial displacements

NOTES:

1) Subsets 9.3.5.2 and 9.3.5.3 must be repeated in turn for every element.
2) Nodes with an initial displacement should be restrained in those particular directions.

---------------+---------------

CARD SET 9.3.5.2
FORMAT : 4I5
NUMBER OF CARDS IN SET : Variable

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	NEASS	The element number with which the element edge is associated
	6-10	NODIS(1)	List of nodal points, in an anticlockwise sequence,
	11-15	NODIS(2)	of the nodes forming the element face on which the
	16-20	NODIS(3)	initial displacement acts

CARD SET 9.3.5.3
INITIAL DISPLACEMENT VALUES
FORMAT : 6F10.0
NUMBER OF CARDS IN SET : Variable

NOTES	COLUMNS	VARIABLES	ENTRY
	1-10	DISX(1)	Value of the X component of the initial displacement of node NODIS(1)
	11-20	DISX(2)	Value of the X component of the initial displacement of node NODIS(2)
	21-30	DISX(3)	Value of the X component of the initial displacement of node NODIS(3)
	31-40	DISY(1)	Value of the Y component of the initial displacement of node NODIS(1)
	41-50	DISY(2)	Value of the Y component of the initial displacement of node NODIS(2)
	51-60	DISY(3)	Value of the Y component of the initial displacement of node NODIS(3)

CARD SET 9.3.6
Maximum Pressure for Crack Pressurization
Only if IMODE(14)=4, in Card 2
FORMAT : 6F10.0
NUMBER OF CARDS IN SET : 1

NOTES	COLUMNS	VARIABLES	ENTRY
	1-10	PMAX	Maximum pressure in crack

CARD SET 10
FLUID FLOW DATA
Only if IMODE(14) = 1, 2, or 3
NUMBER OF CARDS IN SET : Variable

CARD SET 10.1
FLUID CONTROL PARAMETERS
FORMAT : 3I5.3, F10.2

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	NSHELL	Joint cut-off number: highest number of solid material.
	6-10	NJELT	Number of joint elements.
	11-15	NUMFNP	Number of flow nodal points. Number must increase with increasing joint element number.
	16-25	XNHP	Maximum pressure, psi (used for scaling)
	25-35	SPGR	Specific gravity, pound per cubic inch. Put 1.0 since water heads are given in terms of pressure.
	36-45	VISC	Dynamic viscosity, pound sec/inch^2. For water at 20°C, about 1.4 x 10^{-7}.

CARD SET 10.2
FLOW ELEMENTS AND NODES
FORMAT : 3I5

NOTES	COLUMNS	VARIABLES	ENTRY
1	1-5	M	Flow element number. Start with 1, increase with increasing structural joint number.
2	6-10	IXF(M,1)	Nodal point I.
	11-15	IXF(M,2)	Nodal point J.

NOTES:

1) If elements are omitted, the program automatically generates the missing elements and flow nodes.
2) The joint structural element number, IXF(M,3) that corresponds to element M, is automatically generated.

CARD SET 10.3
FLOW NODAL POINT BOUNDARY CONDITIONS
FORMAT : 2I5, 2F10.3

NOTES	COLUMNS	VARIABLES	ENTRY
	1-5	N	Nodal point number.
1	6-10	IB	Number which indicates head or flow rate to be specified. Heads to be input as pressures.
	11-20	XH	Pressure at node N.
	21-30	XQ	Flow rate at Node N.

NOTES:

1) If the number in columns 6-10 is
   - -1 XH is blank and XQ is the specified flow rate.
   - 0 XQ and XH both are blank (corresponding nodal point is an internal point).
   - +1 XQ is blank and XH is the specified head.

```
+------------------------------------------------------------+
|   CARD SET 10.4                                            |
|   APERTURE                                                 |
|   FORMAT : I5                                              |
+------------------------------------------------------------+
```

NOTES	COLUMNS	VARIABLES	ENTRY
1	1-5	NTHICK	Control for aperture distribution

NOTES:

1) NTHICK = 1 gives constant width aperture distribution.
   NTHICK = 2 gives lognormal width aperture distribution.

```
+------------------------------------------------------------+
|   CARD SET 10.4.1                                          |
|   CONSTANT WIDTH APERTURE                                  |
|   Only if NTHICK = 1, in Card 10.4                         |
|   FORMAT : F10.2                                           |
+------------------------------------------------------------+
```

NOTES	COLUMNS	VARIABLES	ENTRY
	1-10	THICKE	Aperture in inches.

```
+------------------------------------------------------------+
|   CARD SET 10.4.2                                          |
|   LOGNORMAL APERTURE DISTRIBUTION                          |
|   Only if NTHICK = 2, in Card 10.4                         |
|   FORMAT : 2F10.2                                          |
+------------------------------------------------------------+
```

NOTES	COLUMNS	VARIABLES	ENTRY
	1-10	AAS	Standard deviation.
	11-20	AAM	Mean.

CARD SET 10.5
MINIMUM APERTURE FOR FLOW
FORMAT : 10.2

NOTES	COLUMNS	VARIABLES	ENTRY
	1-10	WMIN	Minimum aperture for fluid flow. For water, about 10^{-5} inches.

2.2 Restart Capability

A file is specified automatically for dumps at the end of each crack increment cycle. If no crack increment, it occurs after flow and joint iterations have been completed. To restart FEFFLAP simply put the number '99' into the first two columns of the first line of the data set.

3. GENERAL OPERATING INSTRUCTIONS

In this section the procedure for operating FEFFLAP on the Cray computer is described. A sample problem is provided, complete with data set and the teletype prompt and input record. FEFFLAP is a highly interactive code and graphics pages appear on the monitor during execution. These graphics pages are designed to provide the operator with sufficient information at each step so that intelligent decisions can be made. A "mouse" or cursor control is required for the execution of FEFFLAP.

3.1. Graphics Output

The graphics pages that are displayed during the execution of FEFFLAP consist of:

- The finite element mesh - node numbers and/or element numbers can be displayed on the mesh. This is controlled from the input data set.

- Stress plots - one can select all stresses or positive stresses only. Again, this selection occurs in the data input set.

- A plot of all the cracks. An outline of the mesh is provided.

- An amplified distortion plot that is scaled up so that all the larger displacements are easily visible.

- The mesh again; the point of crack initiation or crack propagation is designated by positioning the cursor (or mouse) on that point on the mesh. A desired "window" is selected at this time, again with the aid of the cursor.

- A plot of the new mesh; the new crack increment is included and all the new elements and nodes have been generated automatically. The user now has the option of changing aspect ratios of elements by moving nodes with the aid of the cursor.

- A new stress plot.

- A load-displacement curve for the node selected in the input data set.

- Amplified distortion plots are provided for each iteration on fluid flow or interface nonlinearity.

3.2 Execution Procedure

The preparation of the initial data file to be submitted to FEFFLAP has been described above. Once the program begins execution, a large amount of information is continuously exchanged between the user and the program. Every effort has been made to make the program execution commands as clear and concise as possible. However, there are a few occasions where the user should have a prior knowledge of the underlying assumptions before hitting any key. Those particular cases will be summarized below.

- Each new display page is entered by hitting the RETURN key.

- When the mesh is displayed with 2 boxes (MODIFY and NEXT) on the lower left corner, one can:

  a) Redisplay the mesh by hitting any alphanumeric key while the horizontal cursor is outside those 2 boxes.

  b) Perform local mesh modifications. Drag a node by hitting any alphahumeric key (with the exception of s,S,b,B,r,R,d,D) while the horizontal cursor is on MODIFY. One then puts the cursor on the node to be dragged, hits any key puts the cursor to the new position, and again hits any key.

  c) Improve element aspect ratios by changing the orientation of a side if it is common to 2 triangles. Hit b (or B) while the horizontal cursor is on MODIFY, then position the cursor on the first vertex node of the side, hit any key, position the cursor on the other vertex node, and again hit any key.

d) Resume execution by hitting any key (except b,B,s,S,r,R) while the horizontal cursor is on NEXT.

- When one is asked to select the point of new crack nucleation, one can:

a) Nucleate a crack from a corner node by hitting any key except s,S while the cursors point to the node.

b) Nucleate the crack from an element side by hitting s or S while the cursors point to the location on the side where nucleation is desired.

- The line 'RB, RF, NB, NF, T, E' appears on the teletype when FEFFLAP is initiated. This line can be obtained by typing 'X' after a '?' prompt. The parameters are:

RB - RJET Begin where RJET is the printer fixture
RF - RJET Finish
NB - NIPS Begin
NF - NIPS Finish
T - Time
E - End Program.

For example, typing RB will accumulate all subsequent monitor plots in a file called RAA43B11. Either RF or E closes this file. The hardcopy is then obtained by typing:

```
GIVE RAA43B11 000002 K. END
```

Later files are named RBB43B11, etc. if earlier files are not deleted.

- To reduce clutter due to overrun of element and node numbers in refined areas of a mesh, the user may "zoom" into an area. This is possible anytime a frame is displayed around the mesh. To initiate the zooming process, hit the "r" key and then RETURN. The user will then be prompted for the limits of the rectangle containing the area desired. These limits are expressed as XMIN, YMIN and XMAX, YMAX: these are coordinates of the lower left and upper right vertices of the rectangle, respectively.

- The stress-intensity factors play an important role in the operation of FEFFLAP. The user can obtain a graphical display of the stress-intensity factors on a K_I-K_{II} interaction plane anytime a frame appears around the mesh. This plane will be displayed when the "g" key is hit, followed by a RETURN.

- When a crack is initiated from an interface the following question appears: "What is the other element number?" One must enter zero, i.e., "0".

3.3 Sample Problem

The sample problem is shown in Figure 12. It includes four joints at 24-inch spacing and a 9-inch diameter borehole. Constant, unequal, compressive stresses are applied horizontally and vertically. The borehole is then pressurized to cause fracture initiation and extension. Figure 13 is an amplified picture of the borehole which shows how well 8 finite elements can represent a borehole. This problem utilizes the multiple load capability of FEFFLAP; the field stresses are constant and the pressure in the borehole varies. The following are supplied in sequence:

- the input data set.

- teletype interaction.

- some of the graphics output; fifty-four pictures were generated for this problem, but only a few are shown here for illustration purposes.

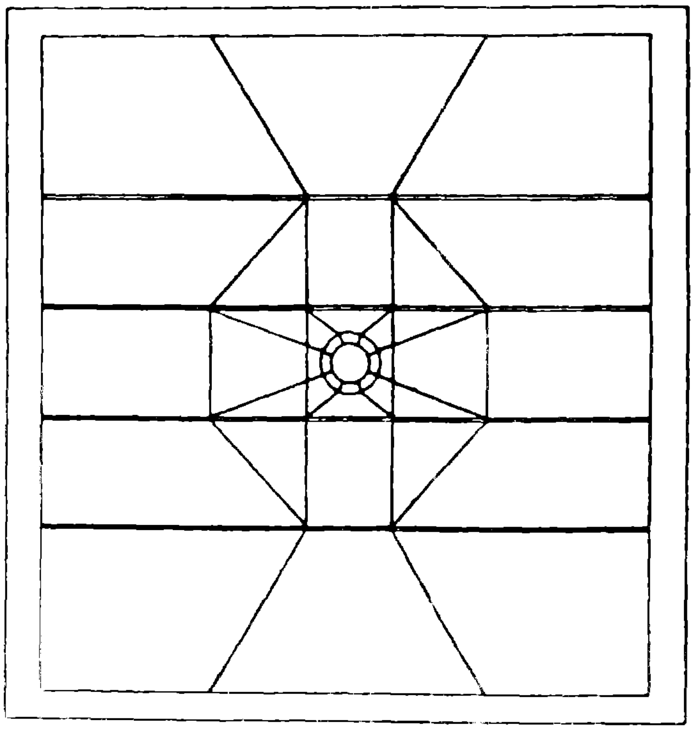


Figure 12. Sample Problem Consisting of a Borehole and Four Joints

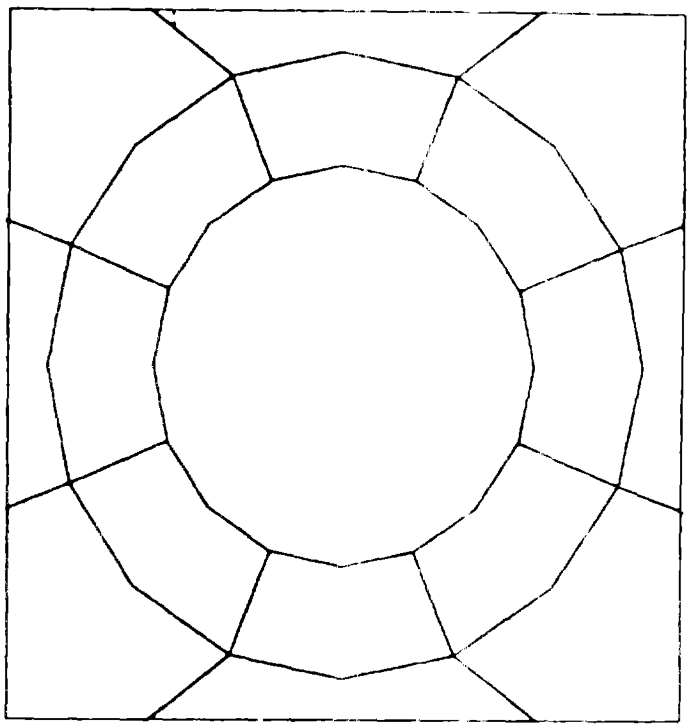


Figure 13. The Borehole Region of Figure 12 Shown Enlarged

3.3.1 Data input file.

CONTROL CARD

ELEMENTS

```
 sample problem: cracks initiating from a borehole in layered media
 188   64   18    4    2    2    9      22111111032001020
1
   1    1    1    2    3   17   28   27   26   16
   2    1    4    5    6   19   31   30   29   18
   3    1    7    8    9   21   34   33   32   20
   4    1   10   11   12   23   37   36   35   22
   5    1   13   14   15   25   66   65  188   24
   6    1   29   30   31   49   39   38
   7    1   32   51   42   41   40   50
   8    1   32   33   34   52   44   43   42   51
   9    1   34   53   46   45   44   52
  10    1   35   36   37   48   47   54
  11    1   26   27   28   58  125  124  123   57
  12    1   29   38   39   60  128  127  126   59
  13    1   40   62   72   71   86   70  129   61
  14    1   40   41   42   63   74   73   72   62
  15    1   42   43   44   64   76   75   74   63
  16    1   44   45   46   65   78   77   76   64
  17    1   46   66  136   67   80   79   78   65
  18    1   80   67  135  134  133   68   82   81
  19    1   84   83   82   68  133  132  131   69
  20    1   86   85   84   69  131  130  129   70
  21    1   72  101   88   87  102  100   86   71
  22    1   72   73   74  182   90   89   88  181
  23    1   74   75   76  183   92   91   90  182
  24    1   76   77   78  184   94   93   92  183
  25    1   78   79   80  185   96   95   94  184
  26    1   80   81   82  186   98   97   96  185
  27    1   82   83   84  187  100   99   98  186
  28    1   84   85   86  180  102  101  100  187
  29    1   47   48   37  120  138  137  136  119
  30    1  188   55   56  122  141  140  139  121
  31    1  126  127  128  143  150  142
  32    1  129  130  131  145  151  144
  33    1  131  132  133  146  153  152  151  145
  34    1  133  134  135  147  153  146
  35    1  136  137  138  149  154  148
  36    1  123  124  125  156  167  166  165  165
  37    1  126  142  150  158  170  169  168  157
  38    1  151  152  153  160  173  172  171  159
  39    1  154  149  138  162  176  175  174  161
  40    1  139  140  141  164  179  178  177  163
  41    2    4   18   29   28   17    3
  42    2   29   59  126  125   58   28
  43    2  126  157  168  167  156  125
  44    2    7   20   32   31   19    6
  45    2   32   50   40   39   49   31
  46    2   40   61  129  128   60   39
  47    2  129  144  151  150  143  128
  48    2  151  159  171  170  158  150
  49    2   10   22   35   34   21    9
  50    2   35   54   47   46   53   34
  51    2   47  119  136  135   66   46
  52    2  136  148  154  153  147  135
  53    2  154  161  174  173  160  153
  54    2   13   24  188   37   23   12
  55    2  188  121  139  138  120   37
  56    2  139  163  177  176  162  138
  57    3  104  103  118  102   87   88
```

58	3	106	105	104	88	89	90
59	3	108	107	106	90	91	92
60	3	110	109	108	92	93	94
61	3	112	111	110	94	95	96
62	3	114	113	112	96	97	98
63	3	116	115	114	98	99	100
64	3	118	117	116	100	101	102

NODAL COORDINATES

1	-72.0	-70.
3	-36.0	-70.
4	-36.0	-70.0
6	-12.0	-70.0
7	-12.0	-70.0
9	12.0	-70.0
10	12.0	-70.0
12	36.0	-70.0
13	36.0	-70.0
15	72.0	-70.0
26	-72.0	-32.0
28	-36.0	-10.0
29	-36.0	-10.0
31	-12.0	-32.0
32	-12.0	-32.0
34	12.0	-32.0
35	12.0	-32.0
37	36.0	-10.0
39	-12.0	-10.0
40	-12.0	-10.0
42	-04.0	-10.0
44	4.0	-10.0
46	12.0	-10.0
47	12.0	-10.0
56	72.0	-32.0
71	-7.0	0.0
72	-6.46716	-2.67878
73	-4.94975	-4.94975
74	-2.67878	-6.46716
75	0.0	-7.0
76	2.67878	-6.46716
77	4.94975	-4.94975
78	6.46716	-2.67878
79	7.0	0.0
80	6.46716	2.67878
81	4.94975	4.94975
82	2.67878	6.46716
83	0.0	7.0
84	-2.67878	6.46716
85	-4.94975	4.94975
86	-6.46716	2.67878
87	-4.5	0.0
88	-4.15746	-1.72208
89	-3.18198	-3.18198
90	-1.72208	-4.15746
91	0.0	-4.5
92	1.72208	-4.15746
93	3.18198	-3.18198
94	4.15746	-1.72208
95	4.5	0.0
96	4.15746	1.72208
97	3.18198	3.18198
98	1.72208	4.15746

```
 99   0.0         4.5
100  -1.72208     4.15746
101  -3.18198     3.18198
102  -4.15746     1.72208
103  -.45         0.0
104  -.415746   -.172208
105  -.318198   -.318198
106  -.172208   -.415746
107   0.0       -.45
108   .172208   -.415746
109   .318198   -.318198
110   .415746   -.172208
111   .45         0.0
112   .415746    .172208
113   .318198    .318198
114   .172208    .415746
115   0.0        .45
116  -.172208    .415746
117  -.318198    .318198
118  -.415746    .172208
123  -72.0       32.0
125  -36.0       10.0
126  -36.0       10.0
128  -12.0       10.0
129  -12.0       10.0
131  -04.0       10.0
133    4.0       10.0
135   12.0       10.0
136   12.0       10.0
138   36.0       10.0
139   36.0       10.0
141   72.0       32.0
160  -12.0       32.0
161  -12.0       32.0
163   12.0       32.0
164   12.0       32.0
165  -72.0       70.0
167  -36.0       70.0
168  -36.0       70.0
170  -12.0       70.0
171  -12.0       70.0
173   12.0       70.0
174   12.0       70.0
176   36.0       70.0
177   36.0       70.0
179   72.0       70.0
180   36.0      -10.0
120 4014
  0    0      -79.0        79.0        -79.0        79.0                GRAPHICS WINDOW
  1
  1   15  179   165
  3    1
  9   24    4
     10.     00.001    60.0           99.0
 41
  1   15  179   165
 32   10                                                                CONSTRAINED NODES
151   11
103   11
104   11  118     1     1
```

```
    1 2090000.0       0.22       1.0                                      400.  MATERIAL PROPERTIES
    1    6000.      400.      400.
 2  1    41000.     3000.      007.5       .01       37.0       1.0   0.0002
 3  1        1.       10.   50000.0       .01       45.0   50000.0    6.000
 4  1    10000.       10.        0.1                                .0000001
insitu stresses
    0     0     1     0                                             APPLIED BOUNDARY LOADS
   16
    1     1     2     3
  1000.0    1000.0    1000.0
    2     4     5     6
  1000.0    1000.0    1000.0
    3     7     8     9
  1000.0    1000.0    1000.0
    4    10    11    12
  1000.0    1000.0    1000.0
    5    13    14    15
  1000.0    1000.0    1000.0
   36   167   166   165
  1000.0    1000.0    1000.0
   37   170   169   168
  1000.0    1000.0    1000.0
   38   173   172   171
  1000.0    1000.0    1000.0
   39   176   175   174
  1000.0    1000.0    1000.0
   40   179   178   177
  1000.0    1000.0    1000.0
    1    26    16     1
  1600.0    1600.0    1600.0
   11   123    57    26
  1600.0    1600.0    1600.0
   36   165   155   123
  1600.0    1600.0    1600.0
    5    15    25    56
  1600.0    1600.0    1600.0
   30    56   122   141
  1600.0    1600.0    1600.0
   40   141   164   179
  1600.0    1600.0    1600.0
  borehole pressure
    0     0     1     0                                             INITIAL BOREHOLE PRESSURE
    8
   21    88    87   102
  4400.0    4400.0    4400.0
   22    90    89    88
  4400.0    4400.0    4400.0
   23    92    91    90
  4400.0    4400.0    4400.0
   24    94    93    92
  4400.0    4400.0    4400.0
   25    96    95    94
  4400.0    4400.0    4400.0
   26    98    97    96
  4400.0    4400.0    4400.0
   27   100    99    98
  4400.0    4400.0    4400.0
   28   102   101   100
  4400.0    4400.0    4400.0
    1    24    28  0000.1      1.0      .000000014                    FLUID FLOW PARAMETERS
```

FLUID FLOW BOUNDARY CONDITIONS

```
  1    1    2
  4    5    6
  9   11   12
 14   17   18
 17   21   22
 24   28   21
  1    1
  2    0
  4    1
  6    0
 10    1
 12    0
 16    1
 18    0
 20    1
 21    1  4400.0    0.0
 28    1  4400.0    0.0
  1
.0002
.000007
```

3.3.2 Teletype interaction.

```
REPAR I=CSAMP2,O=TEST / 5 7
CHANGE DEFAULT I/O PARAMETERS (Y OR N)?

RJET OUTPUT IS ON.
RS,PF,NS,NF,T,E? IRS
BEGIN RJET OUTPUT
?
 MAX CHANGE, IELEM, KS ; OLD,NEW,CHANGE,    THEN KN
   1.378E+03
?
 ?
AMPL. FACTOR   1.3476E+01
?
?
NEW CRACK FACTOR
 3.446E+00
IN ELEMENT   27    NEW CRACK FACTOR IS     3.446E+00
DO YOU WANT A NEW CRACK (YN)?
?N
BY WHAT FACTOR DO YOU WANT TO MODIFY THE LOAD?
FOLLOW WITH A COMMA...
1.
     1.000

   DO YOU WANT TO ITERATE ON FLUID FLOW ?, (Y,N)
?N
```

Interface iterations, no flow

```
  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?Y
 MAX CHANGE, IELEM, KS ; OLD,NEW,CHANGE,    THEN KN
   51    3.000E+03 3.000E+03 0.          4.100E+04 8.512E+06 8.471E+06
          MONITORED NODAL DISPLACEMENTS

 NODE       X-COORD.   Y-COORD.    X-DISPL.      Y-DISP L.      NODE

  134        -8.000    -10.000    -3.3183E-03   1.9014E-02    134
AMPL. FACTOR   3.6628E+01

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?Y
 MAX CHANGE, IELEM, KS ; OLD,NEW,CHANGE,    THEN KN
   51    3.000E+03 3.000E+03 0.          8.512E+06 9.465E+06 9.531E+05
          MONITORED NODAL DISPLACEMENTS


 NODE       X-COORD.   Y-COORD.    X-DISPL.      Y-DISP L.      NODE

  134        -8.000    -10.000    -3.3213E-03   1.9010E-02    134
AMPL. FACTOR   3.6630E+01
```

```
 DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?Y
 MAX CHANGE, IELEM, KS | OLD,NEW,CHANGE,   THEN KN
   51    3.000E+03 3.000E+03 0.          9.465E+06 9.478E+06 1.227E+04
          MONITORED NODAL DISPLACEMENTS

 NODE       X-COORD.   Y-COORD.     X-DISPL.      Y-DISP  L.     NODE

  134        -8.000    -10.000     -3.3214E-03   1.9010E-02    134
AMPL. FACTOR   3.6630E+01

 DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?N
```

Exit interface iteration loop

```
   1.015E+03

??
AMPL. FACTOR   2.7045E+01
?
?R
  INPUT XMIN,YMIN,XMAX,YMAX COMMA DELIMIT
-7.,-7.,7.,7.
?
```

Crack initiation

```
NEW CRACK FACTOR
 2.539E+00
IN ELEMENT   27    NEW CRACK FACTOR IS    2.539E+00
DO YOU WANT A NEW CRACK (YN)?
?Y
 (ELEMENT  27)
DO YOU WANT IT IN ELEMENT SHOWN (YN)?
?N
ENTER ELEMENT # WHERE CRACK IS TO NUCLEATE
FOLLOW WITH A COMMA...
25,
   25
```

Initiation factor (new crack factor) should be unity to start crack

```
 YOU HAVE CHOSEN ELEMENT  25   NEW CRACK FACTOR =    1.419E+00
NEW CRACK FACTOR
 1.419E+00
DO YOU WANT TO CHANGE LOAD TO GET A NEW INITIATION FACTOR?  (Y,N)
?Y
ELEMENT AND GAUSS PT    25    4
STRSS,1 TO 3       5.678E+02    -3.854E+03     1.328E+01


         FOR THE CRACK NUCLEATION
      INCREMENT LOAD FACTOR....................    9.599E-01
      TOTAL LOAD FACTOR........................    9.599E-01

MAX STRESS IN ELEMENT  27    NEW CRACK FACTOR IS    2.142E+00
 YOU STILL HAVE ELEMENT  25   NEW CRACK FACTOR =    1.000E+00
?
NEW CRACK FACTOR
 1.000E+00
```

```
IN ELEMENT   25    NEW CRACK FACTOR IS    1.000E+00
  (ELEMENT  25)
DO YOU WANT IT IN ELEMENT SHOWN (YN)?
?Y
DO YOU WANT TO CHANGE LOAD TO GET A NEW INITIATION FACTOR?  (Y,N)
?N
ENTER (XMIN YMIN) & (XMAX YMAX)
CURSOR INPUT :,3,1
SINGLE CHARACTER:
CURSOR INPUT :222%
SINGLE CHARACTER:
CURSOR INPUT :22 5
SINGLE CHARACTER:
?
ENTER 1 + 2 FOR HOR. + VERT OR 3 FOR COMPUTED DIRECTION OF
?1
DO YOU WANT TO REVERSE THE CRACK DIRECTION (YN)?
?N
```

Use cursor to pinpoint location of crack initiation

```
ENTER LOCATION OF CRACK NUCLEATION
CURSOR INPUT :4!/)
SINGLE CHARACTER: E
ENTER ITS ELEMENT NUMBER
FOLLOW WITH A COMMA...
25,
   25
```

Interface element on borehole perimeter divides due to crack

```
      INTERFACE ELEMENT  61 SPLIT INTO ELEMENTS    61    69
CURSOR INPUT :4! <
SINGLE CHARACTER:
?
   51    3.000E+03 3.000E+03 0.            9.465E+06 9.478E+06 1.227E+04
   5.175E+03
?
BY WHAT FACTOR DO YOU WANT TO MODIFY THE LOAD?
FOLLOW WITH A COMMA...
1.
     1.000
```

Interface iterations, with flow

```
   DO YOU WANT TO ITERATE ON FLUID FLOW ?, (Y,N)
Y
?
 IN FLOW, THE ELEMENT NUMBERS AND OPENINGS, NO FLOW IF LESS THAN  7.0E-06
    41= 2.0E-04    42= 2.0E-04    43= 2.0E-04    44= 2.0E-04    45= 2.0E-04
    46= 2.0E-04    47= 2.0E-04    48= 2.0E-04    49= 2.0E-04    50= 2.0E-04
    51= 2.0E-04    52= 2.0E-04    53= 2.0E-04    54= 2.0E-04    55= 2.0E-04
    56= 2.0E-04    57= 2.0E-04    58= 2.0E-04    59= 2.0E-04    60= 2.0E-04
    61= 2.0E-04    62= 2.0E-04    63= 2.0E-04    64= 2.0E-04    69= 0.
    70= 0.
  IN LODCRK, NJELT =     26 NEDGE(1) =     52
 ITERATION NUMBER     MAX DISPL VECTOR     PERCENT CHANGE
       1                 6.275E-02              6.275E+15
AMPL. FACTOR    1.7881E+01  .

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?Y
 MAX CHANGE, IELEM, KS ; OLD,NEW,CHANGE,    THEN KN
   51    3.000E+03 3.000E+03 0.            9.478E+06 9.293E+06 1.843E+05
```

```
          MONITORED NODAL DISPLACEMENTS

 NODE        X-COORD.  Y-COORD.     X-DISPL.      Y-DISP  L.      NODE

  144        -8.000   -10.000    -3.1645E-03    1.9116E-02     144
AMPL. FACTOR     1.7881E+01

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS  (YN)?
?Y
 MAX CHANGE, IELEM, Ks & OLD,NEW,CHANGE,     THEN Kn
   70     1.000E+01 1.000E+01 0.             1.000E+04 1.000E+01 9.990E+03
          MONITORED NODAL DISPLACEMENTS

 NODE        X-COORD.  Y-COORD.     X-DISPL.      Y-DISP  L.      NODE

  144        -8.000   -10.000    -3.1634E-03    1.9110E-02     144
AMPL. FACTOR     1.7884E+01

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS  (YN)?
?N

   DO YOU WANT TO ITERATE ON FLUID FLOW ?,  (Y,N)
?Y

 IN FLOW, THE ELEMENT NUMBERS AND OPENINGS, NO FLOW IF LESS THAN  7.0E-06
    41= 4.4E-09     42= 5.6E-09     43= 4.4E-09     44=-1.1E-09     45=-4.4E-
    46=-4.4E-09     47=-4.3E-09     48=-1.2E-09     49= 5.9E-09     50=-1.5E-
    51=-3.6E-08     52=-1.6E-08     53= 5.8E-09     54= 5.1E-09     55= 5.5E-
    56= 5.0E-09     57= 1.1E-02     58=-2.7E-03     59=-1.2E-02     60=-9.9E-
    61=-3.6E-03     62= 1.3E-02     63= 2.1E-02     64= 2.0E-02     69= 2.3E-
    70= 3.6E-03
  IN LODCRK, NJELT =       26 NEDGE(1) =      52
 ITERATION NUMBER     MAX DISPL VECTOR     PERCENT CHANGE
        2                    6.269E-02                9.835E-02
AMPL. FACTOR     1.7948E+01

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS  (YN)?
?Y
 MAX CHANGE, IELEM, Ks & OLD,NEW,CHANGE,     THEN Kn
   51     3.000E+03 3.000E+03 0.             9.291E+06 9.251E+06 3.947E+04
          MONITORED NODAL DISPLACEMENTS

 NODE        X-COORD.  Y-COORD.     X-DISPL.      Y-DISP  L.      NODE

  144        -8.000   -10.000    -3.1405E-03    1.8997E-02     144
AMPL. FACTOR     1.7948E+01

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS  (YN)?
?N
```

```
   DO YOU WANT TO ITERATE ON FLUID FLOW ?, (Y,N)
?N

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?N
   8.802E+02
?

?
AMPL. FACTOR   2.7079E+01
?
?
```

Crack extension

```
NEW CRACK FACTOR
 2.201E+00
IN ELEMENT    27    NEW CRACK FACTOR IS     2.201E+00
CRACK EXTENSION FACTOR
 1.872E+01
DO YOU WANT A NEW CRACK (YN)?
?N
DO YOU WANT TO EXTEND A CRACK (YN)?
?Y
SHOW THE CRACK TIP
CURSOR INPUT :4! <
SINGLE CHARACTER:
UPDATED FACTOT=     4.5544E-01     FACLOC =        4.744E-01
ICRAC,OLD ANGLE,ANGLE CHANGE,FAC,FACKKN
   1        7.288E-15    -5.236E-02     1.000E+00      1.600E+01
CRACK EXTENSION FACTOR
 1.000E+00
YOU HAVE CHOSEN CRACK  1
WANT A DIFFERENT TIP?(Y,N)
?N
WANT TO RECALCULATE THE VARIABLE LOAD FACTOR? (Y,N)
?N
?P
   INPUT XMIN,YMIN,XMAX,YMAX COMMA DELIMIT
-13.,-13.,13.,13.
?

  DO YOU WANT TO MODIFY MESH(YN)?
?Y
```

Length of crack increment is changed slightly to result in better mesh

```
WOULD YOU PREFER A DIFFERENT CRACK TIP LOCATION (Y,N)
?Y
INDICATE LAST POINT OF THE NEXT CRACK EXTENTION
CURSOR INPUT :$'+8
SINGLE CHARACTER:
ENTER (XMIN YMIN) & (XMAX YMAX)
CURSOR INPUT :'< 0
SINGLE CHARACTER:
CURSOR INPUT :?079
SINGLE CHARACTER:
```

```
WILL THE CRACK CROSS AN INTERFACE ELEMENT (YN)?

?N
CURSOR INPUT :?0 5
SINGLE CHARACTER:
?
   51     3.000E+03 3.000E+03 0.              9.291E+06 9.251E+06 3.947E+04
   51     3.000E+03 3.000E+03 0.              9.291E+06 9.251E+06 3.947E+04

 DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?

?N
```

Crack closes because no fluid pressure is there yet

```
                                                        *** W A R N I N G ***
CRACK   1 HAS CLOSED: KI = -4.0775E+03 KII =   2.5243E+00

                        KI WILL RESET TO ZERO


   WANT TO CHANGE LOAD FACTOR (YN)? CURRENT COMPUTED LOAD FACTOR=    0.505

?N

  REANALYZE AT NEW LOAD FACTOR (Y,N)?
?N

 DO YOU WANT A STRESS PLOT(YN)?

?Y
   9.096E+01
```

Increase load - load is about half the original borehole pressure

```
?
BY WHAT FACTOR DO YOU WANT TO MODIFY THE LOAD?
FOLLOW WITH A COMMA...
1.5
     1.500
```

New interface iteration, with flow

```
   DO YOU WANT TO ITERATE ON FLUID FLOW ?, (Y,N)
?Y

 IN FLOW, THE ELEMENT NUMBERS AND OPENINGS, NO FLOW IF LESS THAN  7.0E-06
    41= 1.8E-06     42= 2.0E-06     43= 1.8E-06     44= 9.7E-07     45= 1.0E-06
    46= 1.1E-06     47= 9.2E-07     48= 1.0E-06     49= 5.4E-07     50= 4.4E-07
    51= 1.1E-06     52= 8.5E-07     53= 5.6E-07     54= 1.9E-06     55= 2.3E-06
    56= 1.9E-06     57= 7.8E-03     58=-7.1E-03     59=-1.7E-02     60=-1.5E-02
    61=-7.3E-03     62= 1.1E-02     63= 1.9E-02     64= 1.8E-02     69=-1.0E-03
    70= 3.7E-03     72=-1.1E-04     77=-6.2E-04
  IN LOCRK, NJELT =     28 NEDGE(1) =    56
```

```
 ITERATION NUMBER     MAX DISPL VECTOR     PERCENT CHANGE
       1                  6.337E-02              6.337E+15
AMPL. FACTOR    1.1078E+01

   DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?Y
 MAX CHANGE, IELEM, KS ; OLD,NEW,CHANGE,    THEN KN
   51     3.000E+03 3.000E+03 0.          9.251E+06 8.430E+06 8.213E+05
          MONITORED NODAL DISPLACEMENTS

 NODE        X-COORD.  Y-COORD.    X-DISPL.      Y-DISP  L.      NODE

  173        -8.000    -10.000    -2.6600E-03   2.0651E-02     173
AMPL. FACTOR    1.1077E+01

   DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?Y
 MAX CHANGE, IELEM, KS ; OLD,NEW,CHANGE,    THEN KN
   77     1.000E+01 1.000E+01 0.          1.000E+04 1.988E+07 1.987E+07
          MONITORED NODAL DISPLACEMENTS

 NODE        X-COORD.  Y-COORD.    X-DISPL.      Y-DISP  L.      NODE

  173        -8.000    -10.000    -2.6564E-03   2.0624E-02     173
AMPL. FACTOR    1.1086E+01

   DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?N

    DO YOU WANT TO ITERATE ON FLUID FLOW ?, (Y,N)
?Y

 IN FLOW, THE ELEMENT NUMBERS AND OPENINGS, NO FLOW IF LESS THAN  7.0E-06
    41= 1.6E-08     42= 2.1E-08     43= 1.7E-08     44=-3.1E-09     45=-2.4E-08
    46=-2.7E-08     47=-1.8E-08     48=-4.5E-09     49=-7.5E-09     50=-2.5E-07
    51=-4.9E-07     52=-2.7E-07     53=-1.0E-08     54= 3.2E-08     55= 3.7E-08
    56= 3.0E-08     57= 8.3E-03     58=-5.5E-03     59=-1.4E-02     60=-1.2E-02
    61=-6.1E-03     62= 1.2E-02     63= 2.0E-02     64= 1.8E-02     69=-4.8E-05
    70= 9.0E-03     72= 5.3E-03     77= 1.5E-03
  IN LODCRK, NJELT =      28 NEDGE(1) =     56
 ITERATION NUMBER     MAX DISPL VECTOR     PERCENT CHANGE
       2                  6.269E-02              1.069E+00
AMPL. FACTOR    1.0470E+01

   DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?Y
 MAX CHANGE, IELEM, KS ; OLD,NEW,CHANGE,    THEN KN
   77     1.000E+01 1.000E+01 0.          1.988E+07 1.988E+04 1.986E+07
```

```
          MONITORED NODAL DISPLACEMENTS

 NODE        X-COORD.   Y-COORD.     X-DISPL.       Y-DISP  L.      NODE

  173        -8.000    -10.000    -2.4703E-03    1.9167E-02     173
AMPL. FACTOR    1.0452E+01

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS  (YN)?
?Y
 MAX CHANGE, IELEM, KS ; OLD,NEW,CHANGE,    THEN KN
   51     3.000E+03 3.000E+03 0.              7.652E+06 7.591E+06 6.152E+04
          MONITORED NODAL DISPLACEMENTS

 NODE        X-COORD.   Y-COORD.     X-DISPL.       Y-DISP  L.      NODE

  173        -8.000    -10.000    -2.4704E-03    1.9168E-02     173
AMPL. FACTOR    1.0452E+01

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS  (YN)?
?Y
 MAX CHANGE, IELEM, KS ; OLD,NEW,CHANGE,    THEN KN
   51     3.000E+03 3.000E+03 0.              7.591E+06 7.589E+06 2.130E+03
          MONITORED NODAL DISPLACEMENTS

 NODE        X-COORD.   Y-COORD.     X-DISPL.       Y-DISP  L.      NODE

  173        -8.000    -10.000    -2.4704E-03    1.9168E-02     173
AMPL. FACTOR    1.0452E+01

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS  (YN)?
?N

   DO YOU WANT TO ITERATE ON FLUID FLOW ?, (Y,N)
?Y

 IN FLOW, THE ELEMENT NUMBERS AND OPENINGS, NO FLOW IF LESS THAN  7.0E-06
    41=-1.7E-12    42= 2.8E-12    43=-2.7E-12    44= 8.4E-11    45= 3.1E-1
    46=-8.9E-11    47= 1.8E-11    48= 9.6E-11    49=-7.7E-10    50=-2.2E-0
    51=-3.2E-09    52=-2.2E-09    53=-8.1E-10    54= 3.3E-10    55= 4.5E-1
    56= 3.3E-10    57= 7.8E-03    58=-4.7E-03    59=-1.2E-02    60=-9.7E-0
    61=-5.9E-03    62= 1.3E-02    63= 2.0E-02    64= 1.8E-02    69= 1.8E-0
    70= 1.6E-02    72= 1.4E-02    77= 6.5E-03
  IN LODCRK, NJELT =     28 NEDGE(1) =     56
 ITERATION NUMBER     MAX DISPL VECTOR     PERCENT CHANGE
       3                 6.266E-02             4.663E-02
AMPL. FACTOR    1.0452E+01

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS  (YN)?
?Y
 MAX CHANGE, IELEM, KS ; OLD,NEW,CHANGE,    THEN KN
   51     3.000E+03 3.000E+03 0.              7.589E+06 7.589E+06 7.918E+01
```

```
         MONITORED NODAL DISPLACEMENTS

NODE       X-COORD.  Y-COORD.    X-DISPL.       Y-DIFF L.      NODE

 173        -8.000   -10.000    -2.4704E-03    1.9168E-02     173
AMPL. FACTOR    1.0452E+01

 DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?N

  DO YOU WANT TO ITERATE ON FLUID FLOW ?, (Y,N)
?N

 DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?N
```

Exit joint - fluid flow iterations

```
 2.196E+02
?
?
AMPL. FACTOR    2.7139E+01
?R$
?R
 INPUT XMIN,YMIN,XMAX,YMAX COMMA DELIMIT
-40.,-40.,40.,40.
?
```

New increment of crack extension

```
NEW CRACK FACTOR
 5.490E-01
IN ELEMENT   28    NEW CRACK FACTOR IS     5.490E-01
CRACK EXTENSION FACTOR
 1.676E+01
DO YOU WANT A NEW CRACK (YN)?
?N
DO YOU WANT TO EXTEND A CRACK (YN)?
?Y
SHOW THE CRACK TIP
CURSOR INPUT :?0!!
SINGLE CHARACTER:
UPDATED FACTOT=      4.3200E-01     FACLOC =          5.696E-01
ICRAC,OLD ANGLE,ANGLE CHANGE,FAC,FACKKN
   1       -6.402E-02    -8.727E-02      1.005E+00      2.085E+01
CRACK EXTENSION FACTOR
 1.005E+00
YOU HAVE CHOSEN CRACK  1
WANT A DIFFERENT TIP?(Y,N)
?N
WANT TO RECALCULATE THE VARIABLE LOAD FACTOR? (Y,N)
?N
?

 DO YOU WANT TO MODIFY MESH(YN)?

?Y
WOULD YOU PREFER A DIFFERENT CRACK TIP LOCATION (Y,N)
?N
```

```
ENTEF (XMIN YMIN) & (XMAX YMAX)
CURSOR INPUT I'+'I
SINGLE CHARACTER:
CURSOR INPUT IE 7+
SINGLE CHARACTER:

 WILL THE CRACK CROSS AN INTERFACE ELEMENT (YN)?

?Y

 DO YOU WANT THE CRACK TO STOP AT THE                    INTERFACE (YN)?
```

Crack intersects joint system. Joint element is split in two and all flow parameters are automatically adjusted.

```
?Y
 NEW FLOW NODE AT ELEMENT  51 HIGHER FLOW NODE PROPS    INCREMENTED UP
CURSOR INPUT IE  5
SINGLE CHARACTER:
?
   51    3.000E+03 3.000E+03 0.          7.589E+06 7.589E+06 7.918E+01
  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?

?N

   WANT TO CHANGE LOAD FACTOR (YN)? CURRENT COMPUTED LOAD FACTOR=    0.426

?N

  REANALYZE AT NEW LOAD FACTOR (Y,N)?
?N

 DO YOU WANT A STRESS PLOT(YN)?

?Y
   2.410E+02
?
BY WHAT FACTOR DO YOU WANT TO MODIFY THE LOAD?
FOLLOW WITH A COMMA...
1.
     1.000

   DO YOU WANT TO ITERATE ON FLUID FLOW ?, (Y,N)
?Y

 IN FLOW, THE ELEMENT NUMBERS AND OPENINGS, NO FLOW IF LESS THAN  7.0E-06
    41= 5.3E-07    42=-1.4E-06    43=-1.7E-06    44=-4.2E-06    45= 9.9E-06
    46= 6.8E-06    47=-2.8E-06    48= 3.9E-06    49=-2.8E-05    50=-1.7E-04
    51= 1.0E-04    52= 2.2E-05    53= 1.4E-06    54= 1.1E-05    55= 1.3E-05
    56= 1.0E-05    57= 6.3E-03    58=-1.4E-02    59=-2.8E-02    60=-2.1E-02
    61=-5.5E-03    62= 3.4E-03    63= 1.6E-02    64= 1.8E-02    69=-9.7E-03
    70=-1.0E-02    72=-1.1E-02    77=-1.4E-02    87=-2.0E-02    88=-7.4E-03
```

```
  IN LDICPI, NJELT =     30 NEDGE(1) =     60
 ITERATION NUMBER     MAX DISPL VECTOR     PERCENT CHANGE
       1                 7.435E-02                7.435E+15
AMPL. FACTOR   1.8115E+01

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?Y
 MAX CHANGE, IELEM, KS $ OLD,NEW,CHANGE,    THEN KN
   50    3.000E+03 3.000E+03 0.            8.804E+06 1.647E+07 7.666E+06
          MONITORED NODAL DISPLACEMENTS

 NODE       X-COORD.  Y-COORD.    X-DISPL.      Y-DISP L.     NODE

  136        -8.000   -10.000   -2.8747E-03   3.4745E-02    136
AMPL. FACTOR   1.8112E+01

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?N

   DO YOU WANT TO ITERATE ON FLUID FLOW ?, (Y,N)
?N

  DO YOU WANT TO RECOMPUTE THE INTERFACE ELEMENT
 STIFFNESSES AND PERFORM A REANALYSIS (YN)?
?N
   9.556E+02
?
?
AMPL. FACTOR   2.1479E+01
?R
-25.,-25.,25.,25.
  INPUT XMIN,YMIN,XMAX,YMAX COMMA DELIMIT
AMPL. FACTOR   1.0804E+01
?
?X
RB,RF,NB,NF,T,E? :E
EXIT (Y OR N)? :Y

 ALL DONE
```

3.3.3 Some graphics output

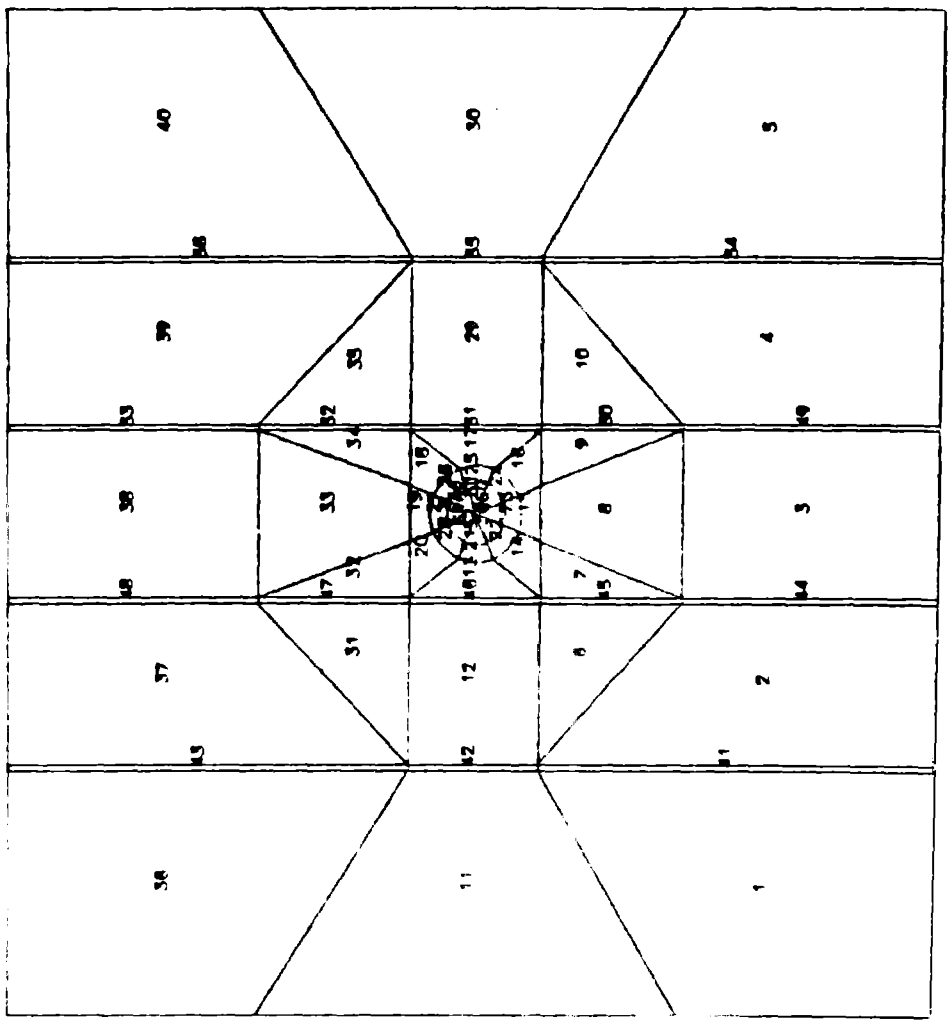


Figure 14. The Initial Mesh with all the Element Numbers.

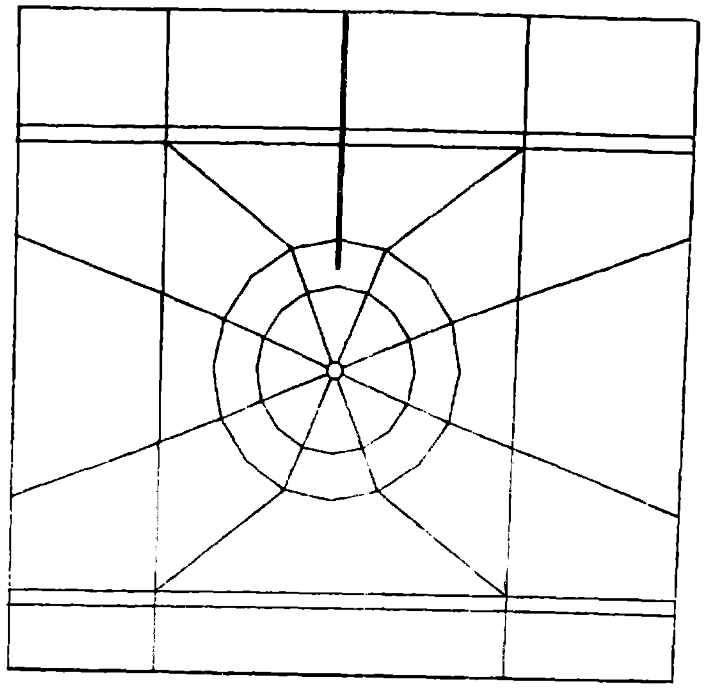


Figure 15. Heavy Line Originates in Element Where Crack is to Initiate and Shows Direction Crack will Follow. Pie Shaped Elements in the Borehole are Interface Elements and are Made very Thick so that they do not Overlap the Mesh when Distorted.

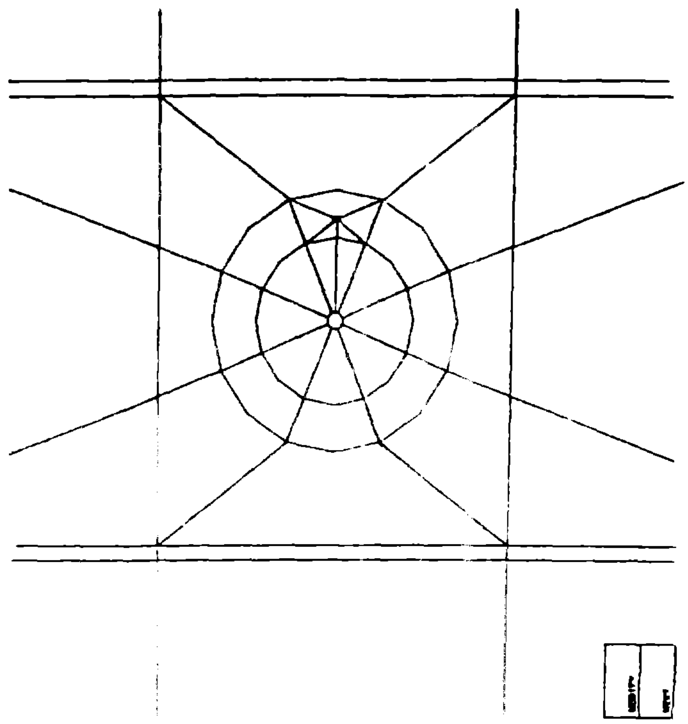


Figure 16. Crack Initiates at Borehole Edge and Ends in Center of Element.

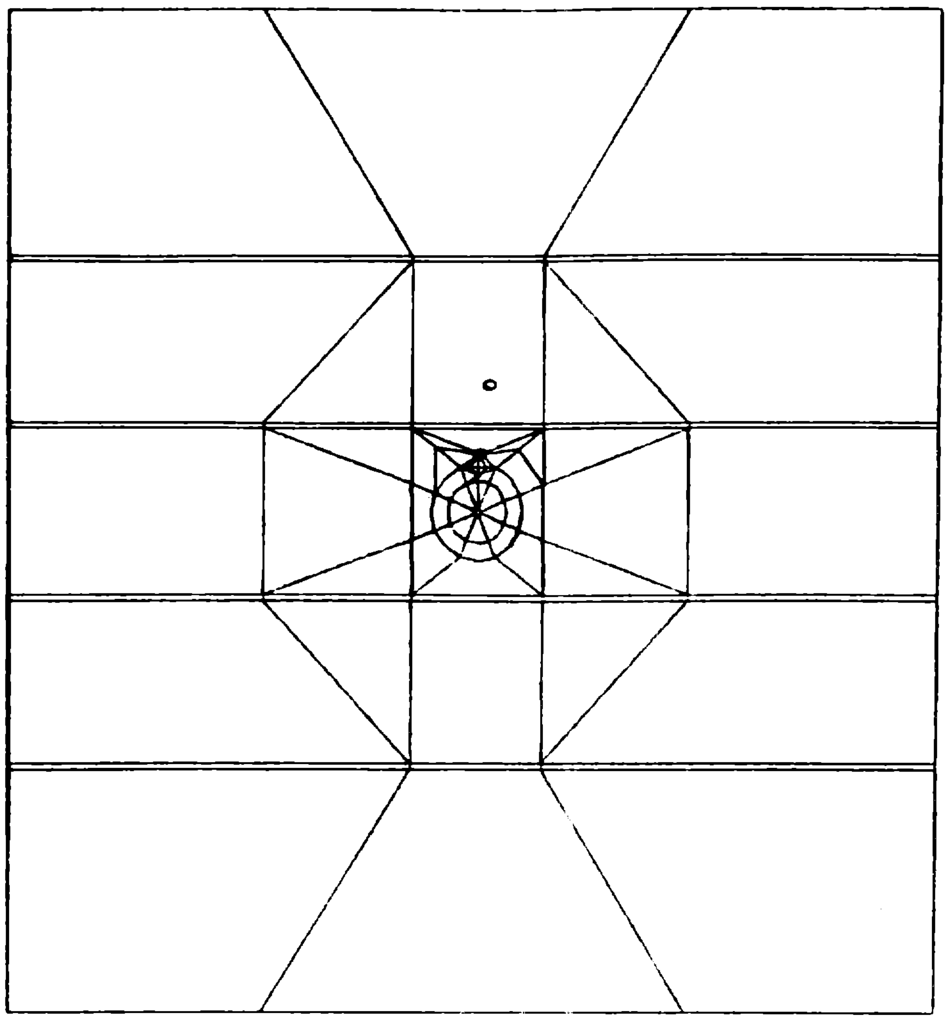


Figure 17. Circle Shows where New Crack Increment will End.

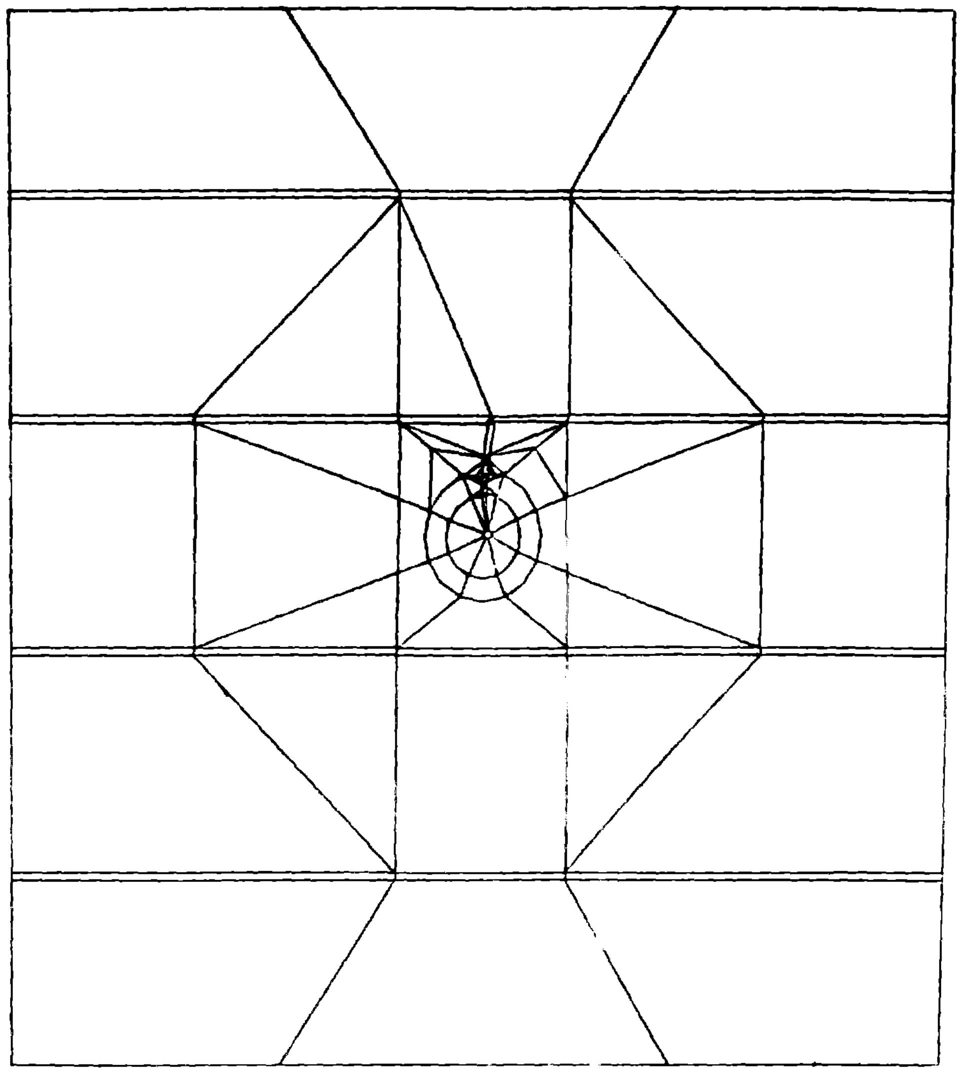


Figure 18. Crack Intersects Joint System.

3.4 Numerical Output

The numerical output file for the sample problem includes:

- A complete reiteration of the input data file including all interpolated nodal positions and all generated nodes and elements. This is indispensable for debugging.
- Stresses in every element (for up to a maximum of 9 Gauss points per element) including magnitude of principal stresses and angle of maximum principal stress with the global x-axis.
- Displacements of every node. Stresses and displacements are provided after each crack increment and after iterations on fluid flow and/or interface nonlinearities.
- Information about crack initiation or extension increments: location of crack, new elements, stress intensity factors, etc.

The output file is not reproduced here because of its size.

4. VERIFICATION AND EXAMPLE PROBLEMS

Solutions to three problems are presented in this section to show the accuracy and versatility of FEFFLAP. In the first problem, the stress intensity factors for two radial cracks from a borehole, as calculated by FEFFLAP, are compared to analytical values. In the second one, FEFFLAP calculations are correlated to results of physical experiments performed on jointed hydrostone blocks. The last problem deals with fracturing of a joint system around a borehole such as may be found in Western gas sands.

4.1 Pressurized Crack and Borehole

FEFFLAP was tested on the cracked borehole problem shown in Figure 19 by calculating Mode I stress intensity factors for two types of loading: a remote biaxial tensile stress, and uniformly pressurized borehole and cracks. The results were compared to established values (15) to obtain an estimate of

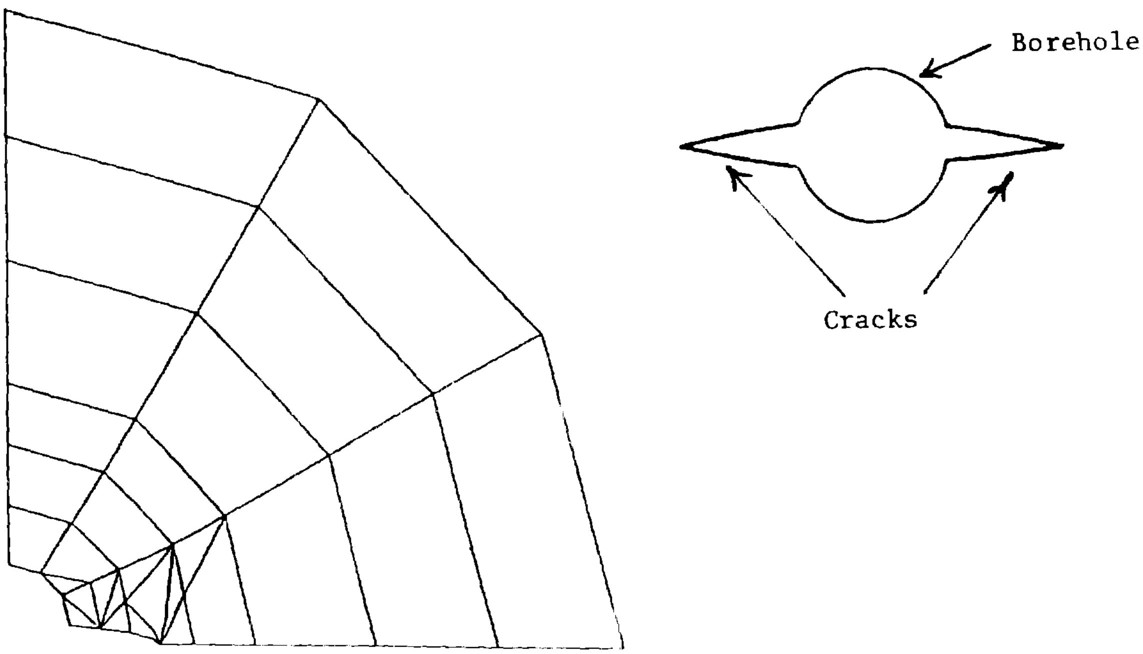


Figure 19: Four-fold Symmetry Finite Element Mesh for a Borehole with Two Opposite Radial Cracks. The Mesh has 28 Elements and 90 Nodes.

FEFFLAP's accuracy. The mesh shown in Figure 19 represents quadrant 1 of the sketch and is all that is required to determine displacements and stresses. For both types of loading each crack length was 1.5 times the borehole radius. The Mode I stress intensity factor calculated in FEFFLAP was 7 percent higher than the established value for both cases. These results are quite good when one considers the coarse finite element mesh. In addition, the mesh is truncated at 10 times the borehole radius while the established values correspond to an infinite medium.

4.2. Hydrostone Block Experiments

Sixteen hydrostone block experiments were performed at LLNL to provide physical test data related to hydrofractures crossing interfaces [16]. The basic test layout is shown in Figure 20. The problem involves two types of hydrostone separated by an interface and also includes the steel platens that are used to load the block. Thus three different solid material types are used. Four joint-interface types are required: (1) the interface between the two hydrostone materials, (2) the interfaces between steel platens and the hydrostone, (3) the joint elements that are inserted into the crack as it propagates, and (4) a set of joint elements around the interior of the borehole, which provides a convenient way to pressurize the hole. The last two joint types are necessary for the fluid flow part of the analysis.

In order to determine the adequacy of FEFFLAP, a 2-D code, to handle the 3-D geometry of Figure 20, the stresses in the mid-vertical section of the block were calculated both with a plane stress FEFFLAP solution, and with a 3-D jointed block code [17]. Results agreed to better than 1%.

Then, two of the tests were analyzed with FEFFLAP using the mesh shown in Figure 21. Figure 22 shows the results of a FEFFLAP analysis of one experiment in which the crack stopped at the interface. Vertical and horizontal loading stresses were 700 and 100 psi, respectively, and the peak pressure in the borehole was 2800 psi. In Figure 23 the FEFFLAP analysis of another experiment shows that a crack reinitiates from the interface. For this case the vertical and horizontal loads were 1800 and 750 psi respectively, and peak borehole pressure was 1700 psi.

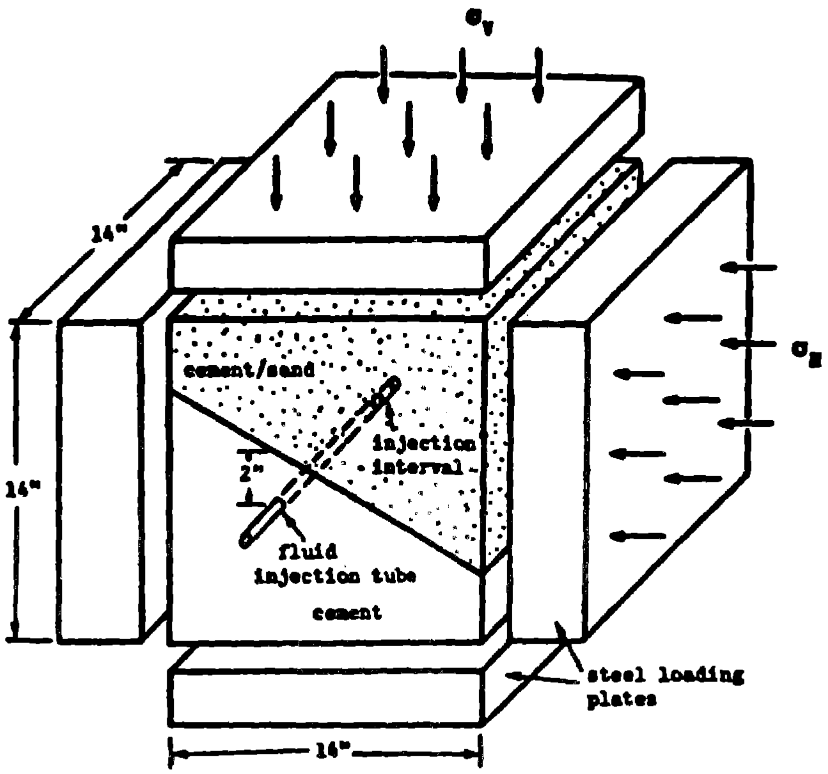


Figure 20: Physical Layout of Jointed Hydrostone Block Experiment [16].

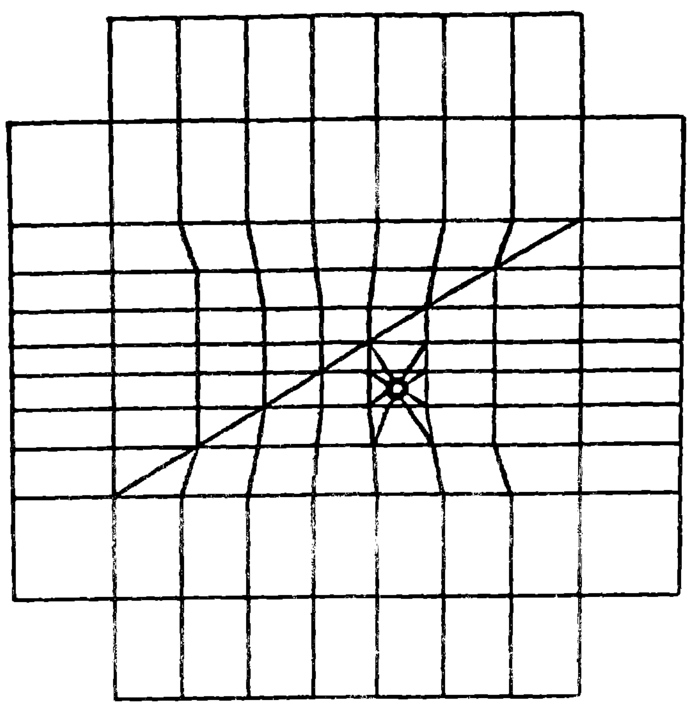


Figure 21: Mesh Used for FEFFLAP Analysis of the Block Tests; the Mesh has 122 solid elements, 46 Joint Elements, and 492 Nodes.

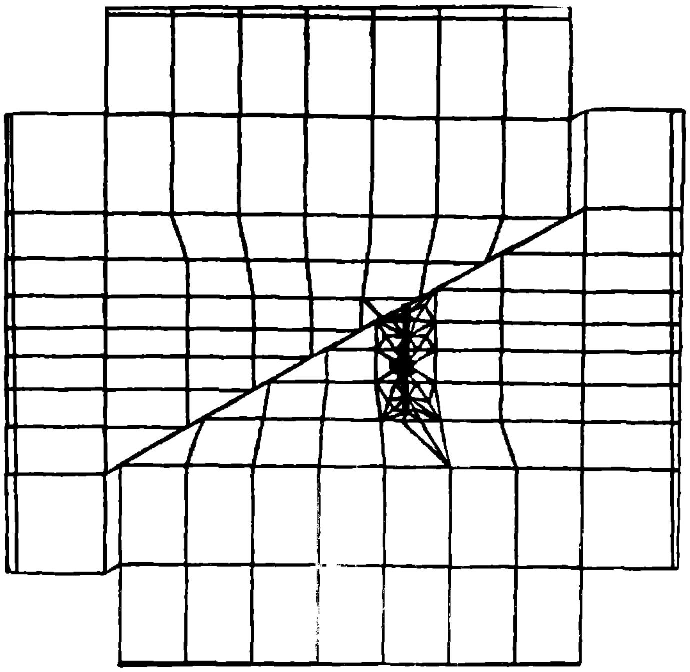


Figure 22: Crack Stops at Interface in Hydrostone Block Experiment and FEFFLAP Analysis.

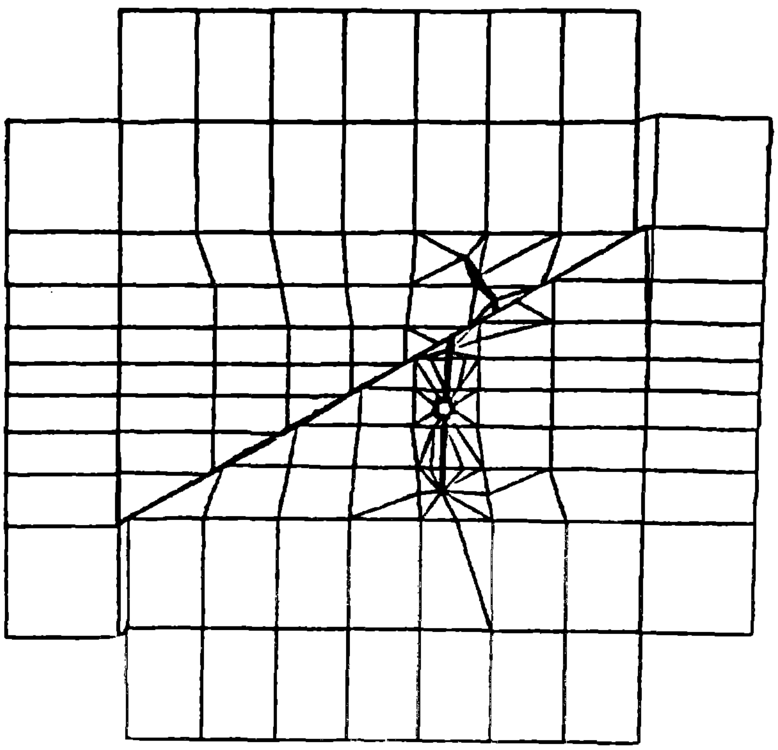


Figure 23: Crack Penetrates Interface in Hydrostone Block Experiment and FEFFLAP Analysis.

4.3 Fracturing and Fluid Flow in a Jointed Network

Figure 24 (half-plane symmetric) shows a borehole in a rock mass with multiple joints. Some of them are continuous throughout the mesh and others are not. The borehole is initially uncracked. The ends of the discontinuous joints are treated as crack tips. The borehole is then pressurized. This initiates a crack. Joint elements are added in the crack as it progresses, to permit flow and pressurization to proceed. Figure 25 shows that as the crack nears the first interface (the first joint) it opens it up by inducing a tensile stress field ahead of the crack front. To our knowledge this interface opening has not previously been numerically simulated. The crack propagates to the first rock joint and the fluid flow opens that joint. Figure 26 shows a close-up of the steady state reached: the crack did not cross the first joint, and all the flow goes to open other existing joints. Figure 27 shows the resulting grid. To our knowledge, this constitutes the first demonstration of a capability combining discrete crack propagation and fracture flow in jointed media.

5. SUMMARY

The stimulation of complex gas reservoirs is best done by massive fracturing. The fractures are driven by fluids such as gels and foams. The prediction of fracture extent requires very sophisticated models, to account for the real geologies in which induced fractures interact with natural discontinuities.

We have developed a state-of-the-art model to describe fluid-driven fracture propagation in naturally jointed gas-bearing rock formations. It is a finite element code, named FEFFLAP (Finite Element Fracture and Flow Analysis Program). The program is highly interactive, with extensive graphical displays of the fracture behavior. Many automatic features for input generation, zoning, and rezoning make the code particularly efficient. The fracture mechanics, solid mechanics, and fluid mechanics are fully coupled.

Model verification has been performed against analytical solutions and physical experiments. The program was developed on a CRAY computer and can be transcoded for use on workstations and minicomputers. This document constitutes the user's manual for the code and provides sample problems used for verification and demonstration of the code's versatility.

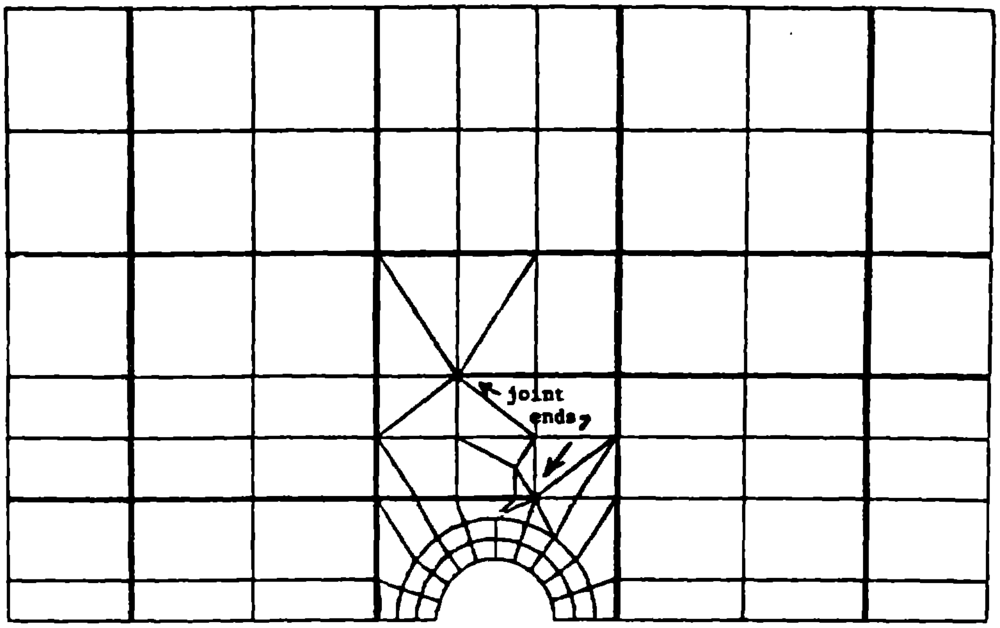


Figure 24: Borehole in a Jointed Rock Mass. Joints are Shown as Heavy Lines. Note That Two of Them are Not Continuous.

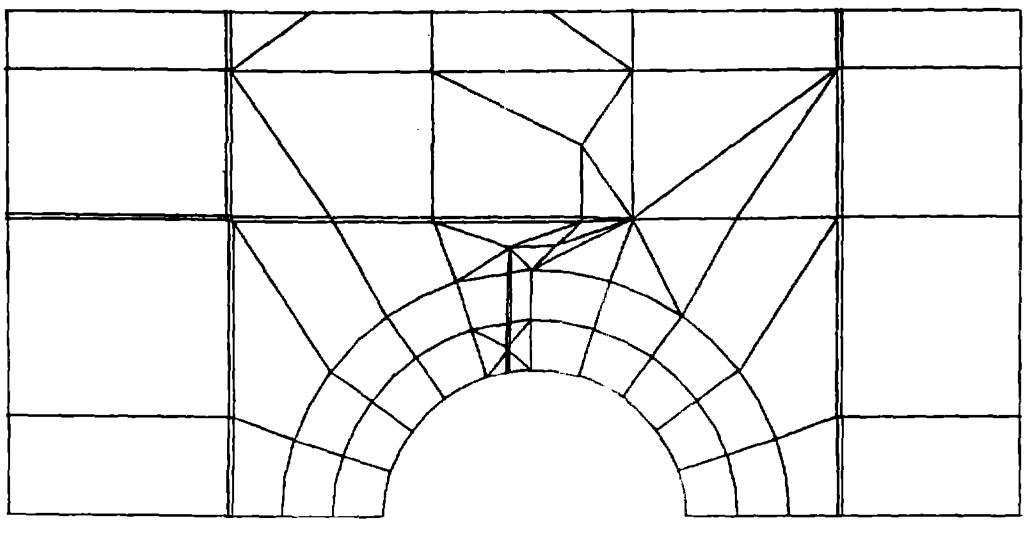


Figure 25: Crack Approaching the Nearest Joint Opens the Interface by the Tensile Stresses Ahead of the Crack Front.

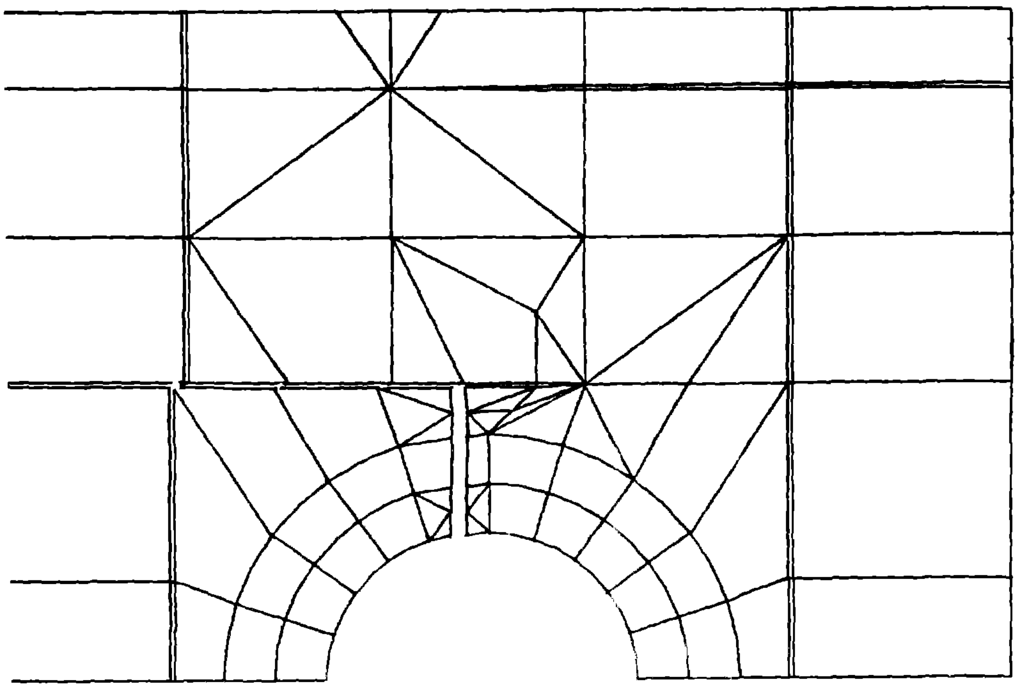


Figure 26: The Crack has Reached the Nearest Interface.

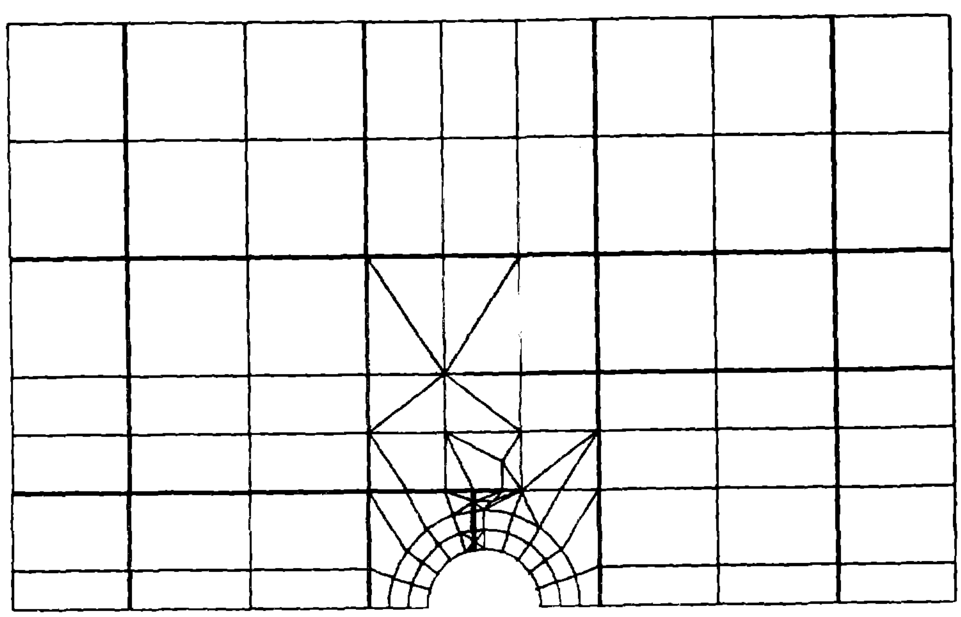


Figure 27: The Resulting Grid.

6. REFERENCES

1. Ingraffea, A.R., "Discrete Fracture Propagation in Rock, Laboratory Tests and Finite Element Analysis", Ph.D. Dissertation, Civil Engineering, University of Colorado, Boulder, 1977.

2. Ingraffea, A.R., and Heuze, F.E., "Finite Element Models for Rock Fracture Mechanics", Int. J. for Numerical and Analytical Methods in Geomechanics, vol. 4, pp. 25-43, 1980.

3. Ingraffea, A.R. and Saouma, V., "Numerical Modeling of Discrete Crack Propagation in Reinforced and Plain Concrete", Application of Fracture Mechanics to Concrete Structures, G.C. Sih and A. DiTommaso editors, Martinus Nijhoff Publishers, 1984.

4. Noorishad, J., Witherspoon, P.A., and Brekke, T.L., "A Method for Coupled Stress Flow and Analysis of Fractured Rock Masses", Publ. No. 71-6, Civil Engineering, University of California, Berkeley, March, 1971.

5. Heuze, F.E., "JPLAXD: A Finite Element Program for Static Plane and Axisymmetric Analysis of Structures in Jointed Rock", Lawrence Livermore National Laboratory, UCID-19047, 1981.

6. Freese, C. E., and Tracey, D. M., "The Natural Isoparametric Triangle Versus the Collapsed Quadrilateral for Elastic Crack Analysis", Int. J. of Fracture, Vol. 12, p. 767, 1976.

7. Barsoum, R.S., "On the Use of Isoparametric Finite Elements in Linear Fracture Mechanics", Int. J. Num. Meth. in Engrg., Vol. 10, No. 1, pp. 25-37, 1976.

8. Gibbs, N.E., Poole, W.J., and Stockmeyer, P.K., "An Algorithm for Reducing the Bandwidth and Profile of a Sparse Matrix", SIAM J. Num. Anal., Vol. 13, pp 236-250, 1976.

9. Erdogan, F. and Sih, G.C., "On the Crack Extension in Plates Under Plane Loading and Transverse Shear", ASME, J. Basic Engr., Vol. 85, pp. 519-527, 1963.

10. Hussain, M.A., Pu, S.L., and Underwood, J.H., "Strain Energy Release Rate for a Crack Under Combined Mode I and Mode II", Fracture Analysis, ASTM, STP 560, pp. 2-28, 1974.

11. Sih, G.C., "Strain-Energy-Density Factor Applied to Mixed-Mode Crack Problems", Int. J. Fract. Mech., Vol. 10, No. 3, pp. 305-321 1974.

12. Shih, C.F., Delorenzi, H.G., and German, M.D., "Crack Extension Modeling with Singular Quadratic Isoparametric Elements", Int. J. Fract., Vol. 12, pp. 647-651, 1976.

13. Ingraffea, A.R., Gerstle, W., Gergely, P., and Saouma, V., "Fracture Mechanics of Bond in Reinforced Concrete", J. of Structural Div., ASCE, J. Struct. Engr., Vol. 110, No. 4, Apr. pp 871-890, 1984.

14. Fenwick, R.C. and Paulay, T., "Mechanics of Shear Resistance of Concrete Beams", J. Struct. Div., ASCE, Vol. 94, No. ST10, pp. 2325-2350, 1968.

15. Sih, G.C., "Handbook of Stress Intensity Factors", Institute of Fracture and Solid Mechanics, Lehigh Univ., Bethlehem, PA, p. 1.2.8.

16. Thorpe, R.K., F.E. Heuze, and R.J. Shaffer, "An Experimental Study of Hydraulic Fracture-Interface Interaction", UCID-20114, Lawrence Livermore National Laboratory, July, 1984.

17. Shaffer, R.J., Thorpe, R.K., Ingraffea, A.R., and Heuze, F.E., "Numerical and Physical Studies of Fluid-Driven Fracture Propagation in Jointed Rock", Proc. SPE/DOE/GRI Unconv. Gas Recovery Symp., Pittsburgh, PA, pp. 471-482, May, 1984. Also Lawrence Livermore National Laboratory, UCRL-90071.

7. ACKNOWLEDGMENTS

This report was prepared by the LLNL Unconventional Gas Program for Morgantown Energy Technology Center, Morgantown, WV, under Contract W-7405-ENG-48, with the U.S. Department of Energy.

The authors are grateful to L. Burrow for her excellent typing of the manuscript. We also acknowledge the fine contribution of R. Thorpe, LLNL, to the analysis of the hydrostone block experiments.